



# **NAVAL POSTGRADUATE SCHOOL**

**MONTEREY, CALIFORNIA**

## **THESIS**

**EXPERIMENTAL INVESTIGATION AND NUMERICAL  
PREDICTION OF THE PERFORMANCE OF A CROSS-  
FLOW FAN**

by

Huai-Te Yu

December 2006

Thesis Advisor:  
Second Reader:

Garth V. Hobson  
Raymond P. Shreeve

**Approved for public release; distribution is unlimited**

THIS PAGE INTENTIONALLY LEFT BLANK

<b>REPORT DOCUMENTATION PAGE</b>			<i>Form Approved OMB No. 0704-0188</i>	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instruction, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188) Washington DC 20503.				
<b>1. AGENCY USE ONLY (Leave blank)</b>		<b>2. REPORT DATE</b> December 2006	<b>3. REPORT TYPE AND DATES COVERED</b> Master's Thesis	
<b>4. TITLE AND SUBTITLE</b> Experimental Investigation and Numerical Predication of a Cross-Flow Fan			<b>5. FUNDING NUMBERS</b>	
<b>6. AUTHOR(S)</b> Yu, Huai-Te				
<b>7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)</b> Naval Postgraduate School Monterey, CA 93943-5000			<b>8. PERFORMING ORGANIZATION REPORT NUMBER</b>	
<b>9. SPONSORING /MONITORING AGENCY NAME(S) AND ADDRESS(ES)</b> N/A			<b>10. SPONSORING/MONITORING AGENCY REPORT NUMBER</b>	
<b>11. SUPPLEMENTARY NOTES</b> The views expressed in this thesis are those of the author and do not reflect the official policy or position of the Department of Defense or the U.S. Government.				
<b>12a. DISTRIBUTION / AVAILABILITY STATEMENT</b> Approved for public release; distribution is unlimited			<b>12b. DISTRIBUTION CODE</b>	
<b>13. ABSTRACT</b>  <p>The concept of a fan-wing aircraft configuration for the purpose of vertical takeoff and landing has drawn much attention. Recently, more investigations revealed that a cross-flow fan (CFF) was capable of providing the propulsion. Several characteristics of the off-design performance of a CFF were experimentally measured, but insufficient numerical predictions were obtained.</p> <p>In the present study, the commercial CFD software ANSYS CFX was employed to calculate the unsteady flow through a CFF with a sliding mesh incorporated. The results of the CFD showed the necessity to re-investigate the cross-flow fan with 12-inch diameter, 1.5-inch span and 30 blades, and additional measurement locations were implemented to carry out a more accurate experiment. A new digital sensor array was used to record the pressures within the experiment, which contributed to the high fidelity of the present data. Successful comparisons were made between the predicted and measured performance at various rotational speeds from an open throttle position to a setting at stall. Visualization of the computed flow field showed where stall occurred, both within the rotor and in the exhaust duct.</p>				
<b>14. SUBJECT TERMS</b> Cross-Flow Fan, Computational Fluid Dynamics (CFD), vertical and short takeoff and landing (V/STOL)			<b>15. NUMBER OF PAGES</b> 131	
			<b>16. PRICE CODE</b>	
<b>17. SECURITY CLASSIFICATION OF REPORT</b> Unclassified	<b>18. SECURITY CLASSIFICATION OF THIS PAGE</b> Unclassified	<b>19. SECURITY CLASSIFICATION OF ABSTRACT</b> Unclassified	<b>20. LIMITATION OF ABSTRACT</b> UL	

THIS PAGE INTENTIONALLY LEFT BLANK

**Approved for public release; distribution is unlimited**

**EXPERIMENTAL INVESTIGATION AND NUMERICAL PREDICTION OF  
THE PERFORMANCE OF A CROSS-FLOW FAN**

Huai-Te Yu  
Captain, Taiwan Army  
B.S., Chung Cheng Institute of Technology, 2000

Submitted in partial fulfillment of the  
requirements for the degree of

**MASTER OF SCIENCE IN MECHANICAL ENGINEERING**

from the

**NAVAL POSTGRADUATE SCHOOL  
December 2006**

Author: Huai-Te Yu

Approved by: Garth V. Hobson  
Thesis Advisor

Raymond P. Shreeve  
Second Reader

Anthony J Healey  
Chairman, Department of Mechanical and Astronautical  
Engineering

THIS PAGE INTENTIONALLY LEFT BLANK

## **ABSTRACT**

The concept of a fan-wing aircraft configuration for the purpose of vertical takeoff and landing has drawn much attention. Recently, more investigations revealed that a cross-flow fan (CFF) was capable of providing the propulsion. Several characteristics of the off-design performance of a CFF were experimentally measured, but insufficient numerical predictions were obtained.

In the present study, the commercial CFD software ANSYS CFX was employed to calculate the unsteady flow through a CFF with a sliding mesh incorporated. The results of the CFD showed the necessity to re-investigate the cross-flow fan with 12-inch diameter, 1.5-inch span and 30 blades, and additional measurement locations were implemented to carry out a more accurate experiment. A new digital sensor array was used to record the pressures within the experiment, which contributed to the high fidelity of the present data. Successful comparisons were made between the predicted and measured performance at various rotational speeds from an open throttle position to a setting at stall. Visualization of the computed flow field showed where stall occurred, both within the rotor and in the exhaust duct.

THIS PAGE INTENTIONALLY LEFT BLANK



# TABLE OF CONTENTS

<b>I.</b>	<b>INTRODUCTION.....</b>	<b>1</b>
<b>II.</b>	<b>EXPERIMENTAL SETUP .....</b>	<b>3</b>
<b>A.</b>	<b>DESCRIPTION OF THE EXPERIMENTAL APPARATUS.....</b>	<b>3</b>
1.	Test Rig .....	3
2.	Cross-Flow Fan Test Assembly (CFFTA).....	3
3.	Control Station .....	5
4.	Data Acquisition and Instrumentation .....	5
<b>B.</b>	<b>DATA ACQUISITION AND REDUCTION.....</b>	<b>7</b>
<b>C.</b>	<b>TEST PLANS .....</b>	<b>11</b>
<b>III.</b>	<b>DESCRIPTION OF THE NUMERICAL MODEL.....</b>	<b>13</b>
<b>A.</b>	<b>OVERVIEW .....</b>	<b>13</b>
<b>B.</b>	<b>GEOMETRY AND GRID GENERATION .....</b>	<b>13</b>
<b>C.</b>	<b>METHODOLOGY AND BOUNDARY CONDITIONS.....</b>	<b>15</b>
<b>D.</b>	<b>SIMULATION PLANS .....</b>	<b>17</b>
<b>IV.</b>	<b>RESULTS AND DISCUSSION .....</b>	<b>19</b>
<b>A.</b>	<b>MESH SENSITIVITY STUDY .....</b>	<b>19</b>
<b>B.</b>	<b>TIME-STEP SELECTION .....</b>	<b>19</b>
<b>C.</b>	<b>INITIAL SIMULATION RESULTS .....</b>	<b>20</b>
1.	Convergence History .....	20
2.	Flow Visualization.....	22
<b>D.</b>	<b>EXPERIMENTAL DATA CONSISTENCY STUDY .....</b>	<b>26</b>
1.	Test Plan I.....	27
2.	Test Plan II .....	28
<b>E.</b>	<b>PERFORMANCE OF THE BASELINE CONFIGURATION AT VARIOUS ROTATIONAL SPEEDS.....</b>	<b>30</b>
<b>F.</b>	<b>OFF-DESIGN PERFORMANCE AT VARIOUS ROTATIONAL SPEEDS .....</b>	<b>34</b>
<b>V.</b>	<b>SUMMARY AND CONCLUSIONS .....</b>	<b>43</b>
<b>APPENDIX A.</b>	<b>CFF_DAQ STRUCTURAL TREE .....</b>	<b>45</b>
<b>APPENDIX B.</b>	<b>MASS FLOW RATE MEASUREMENT OF THE CFF .....</b>	<b>47</b>
<b>APPENDIX C.</b>	<b>UNCERTAINTY STUDY .....</b>	<b>49</b>
<b>APPENDIX D.</b>	<b>GEOMETRY GENERATION IN ICEM-CFD .....</b>	<b>53</b>
<b>APPENDIX E.</b>	<b>PROCEDURES OF SETUP IN CFX-PRE.....</b>	<b>99</b>
	<b>LIST OF REFERENCES .....</b>	<b>113</b>
	<b>INITIAL DISTRIBUTION LIST .....</b>	<b>115</b>

THIS PAGE INTENTIONALLY LEFT BLANK

## LIST OF FIGURES

Figure 1.	Schematic of the Turbine Test Rig (from Ref 5) .....	3
Figure 2.	Cross-Flow Test Assembly (CFFTA, from Ref. 5) .....	5
Figure 3.	Combination probes and pressure tap layout .....	6
Figure 4.	CFF_DAQ graphical user interface .....	7
Figure 5.	Schematic of the position of the combination probes in the exhaust duct.....	9
Figure 6.	CFFTA for (a) Test Plan I and (b)Test Plan II.....	12
Figure 7.	Identification of the ten domains of the CFF mesh.....	14
Figure 8.	The planes between the interfaces .....	15
Figure 9.	The meshes around blades .....	15
Figure 10.	Geometric profile of (a) Simulation Plan I and (b)Simulation Plan II .....	18
Figure 11.	Convergence history of mass flow rate at 3000RPM .....	21
Figure 12.	Convergency history of total pressure ratio and total temperature ratio at 3000RPM .....	21
Figure 13.	Convergency history of efficiency at 3000RPM .....	22
Figure 14.	Vector plot of velocity in HPC and LPC at 3000RPM.....	23
Figure 15.	Distribution of total gauge pressure at 3000RPM.....	24
Figure 16.	Distribution of total temperature at 3000RPM .....	25
Figure 17.	Distribution of Mach number at 3000RPM .....	26
Figure 18.	Consistency study of total pressure versus mass flow rate for Test Plan I.....	27
Figure 19.	Consistency study of efficiency versus mass flow rate for Test Plan I .....	28
Figure 20.	Consistency study of thrust versus mass flow rate for Test Plan I .....	28
Figure 21.	Consistency study of thrust versus power for Test Plan I.....	28
Figure 22.	Consistency study of total pressure ratio versus mass flow rate for Test Plan II.....	29
Figure 23.	Consistency study of efficiency versus mass flow rate for Test Plan II .....	29
Figure 24.	Consistency study of thrust versus mass flow rate for Test Plan II.....	30
Figure 25.	Consistency study of thrust versus power for Test Plan II .....	30
Figure 26.	Comparison of total pressure ratio versus mass flow rate for Test Plan I .....	31
Figure 27.	Comparison of total temperature ratio versus mass flow rate for Test Plan I.....	32
Figure 28.	Comparison of efficiency versus mass flow rate for Test Plan I .....	32
Figure 29.	Comparison of thrust versus mass flow rate for Test Plan I.....	33
Figure 30.	Comparison of power versus mass flow rate for Test Plan I .....	33
Figure 31.	Comparison of thrust versus power for Test Plan I .....	34
Figure 32.	Comparison of total pressure ratio versus mass flow rate for Test Plan II.....	35
Figure 33.	Comparison of total temperature ratio versus mass flow rate for Test Plan II.....	35
Figure 34.	Comparison of efficiency versus mass flow rate for Test Plan II.....	36
Figure 35.	Comparison of thrust versus mass flow rate for Test Plan II.....	36
Figure 36.	Comparison of power versus mass flow rate for Test Plan II.....	37
Figure 37.	Comparison of thrust versus power for Test Plan II.....	37
Figure 38.	Combination plot of Test Plan I and Test Plan II for total pressure ratio versus mass flow rate .....	38

Figure 39.	Combination plot of Test Plan I and Test Plan II for total temperature ratio versus mass flow rate .....	39
Figure 40.	Combination plot of Test Plan I and Test Plan II for efficiency versus mass flow rate .....	39
Figure 41.	Comparison of experimental and computed results for total pressure ratio versus mass flow rate .....	41
Figure 42.	Comparison of experimental and computed results for total temperature ratio versus mass flow rate.....	41
Figure 43.	Comparison of experimental and computed results for efficiencies versus mass flow rate .....	42
Figure 44.	Distribution of Mach at 3000RPM with stall position setting .....	43

## LIST OF TABLES

Table 1.	Thermocouple scanning multiplexer channel assignments.....	6
Table 2.	Scanivalve port assignments .....	7
Table 3.	Area values of each zone .....	9
Table 4.	Summary of simulation plans .....	17

THIS PAGE INTENTIONALLY LEFT BLANK

## **ACKNOWLEDGMENTS**

I would like to express my appreciation to the following people whose valuable contribution to the completion of this thesis.

Professor Garth Hobson, for sparking my interest in the cross-flow fan concept, and without his patience and instruction, I would not have come this far.

Professor Max Platzner, for his enthusiasm and guidance into this topic. Without his hypercritical examination into the model, my simulation could not be run so smoothly

Professor Ray Shreeve, for his instruction in a course of compressible flow. He not only is an excellent instructor, but showed the utmost solicitude to me.

Dr. Anthony Gannon, for his assistance in programming the data acquisition system. With his understanding and explanation, the program was created in just few days.

Rick Still and John Gibson, for always making the test rig run properly. Their reliability made the experiment a success.

Finally, I would like to thank my wife, Jui-Feng Tsou, for her consideration. With her full support, I can concentrate on this study with no second thought. Meanwhile, we are glad to have my first son, Jason Yu.

THIS PAGE INTENTIONALLY LEFT BLANK



## I. INTRODUCTION

Vertical and short takeoff and landing (V/STOL) vehicles have been studied for many years. Requirements for larger thrust and higher fuel efficiency were encountered. In order to meet those requirements, a device, the Cross-Flow Fan (CFF), was first introduced in 1975 by the Vought Systems Division (VSD) [Ref. 1] of LTV Aerospace Corporation. Several tests were performed on various fan and housing configurations. Also, the location of shocks and vortices within the device were investigated. The capabilities of this device to be used as a propulsion system were recommended. However, more experiments needed to be carried out to provide more understanding of the off-design performance of the cross-flow fan.

Based on the experimental data by VSD, Dean H. Gossett [Ref. 2] designed a light-weight VTOL aircraft to be used for transportation between cities. Fan number 6 with an exit duct height of 4.6-inch was selected due to its higher thrust production. Theoretically, his concept of a wing-and-canard type air vehicle sparked the possibilities of utilizing a cross-flow fan as the sole propulsive device for the aircraft.

The performance of a cross-flow fan was tested in the Turbopropulsion Laboratory of the Naval Postgraduate School by M. Scot Seaton [Ref. 3]; and a numerical model, using FLO++, was also initiated to predict the performance for further design purposes. Moreover the concept of the Fan-Wing aircraft was not only proposed but also treated numerically in a conceptual wing configuration. This work was continued by Wee Teck Cheng [Ref. 4] who measured the off-design performance of the 12-inch CFF up to 6000RPM. He also predicted the design performance of the CFF at 3000RPM using FLO++, however, his off-design prediction did not follow the experimental trends.

Using the experimental results from [Ref. 4], a cross-flow fan embedded into an airfoil was proposed and designed by Joseph D. Kummer [Ref. 5]; an incredible enhancement of airfoil lift coefficient under specific conditions was predicted by employing the software package, FLUENT. Practically, it would be a problem to couple this device into an aircraft. It was revealed that the utilization of a cross-flow fan to achieve high lift propulsion should be given more attention.

Since the concept of a Fan-Wing aircraft with a cross-flow fan was regarded as a possible application, a smaller sized cross-flow fan was needed and its performance needed to be investigated. More recently, a 6-inch CFF was tested both with a 1-1/2-inch span and a 6-inch span rotor at low to moderate speeds [Ref. 6]. Since the simulation by FLO++ only provided results for incompressible flow, the software package, ANSYS CFX, was used in the present study. The code could more accurately model compressible flow and could also model rotating machinery. Discrepancies found in the measurement of mass flow rate in the previous data, necessitated repeating the test of the 12-inch rotor with a new pressure measuring digital sensor array. The off-design performance prediction using CFX-5 more closely followed the experimental results obtained in the present study.

## II. EXPERIMENTAL SETUP

### A. DESCRIPTION OF THE EXPERIMENTAL APPARATUS

#### 1. Test Rig

The fan was tested in the Turbine Test Rig (TTR) at the Turbopropulsion Laboratory of the Naval Postgraduate School. The TTR was powered by compressed air from an Allis-Chalmers compressor, which was driven by a 1,250 horsepower motor. The compressor could produce 10,000 cubic feet per minute of air at a maximum pressure of 30 psig. A schematic of TTR is shown in Figure 1.

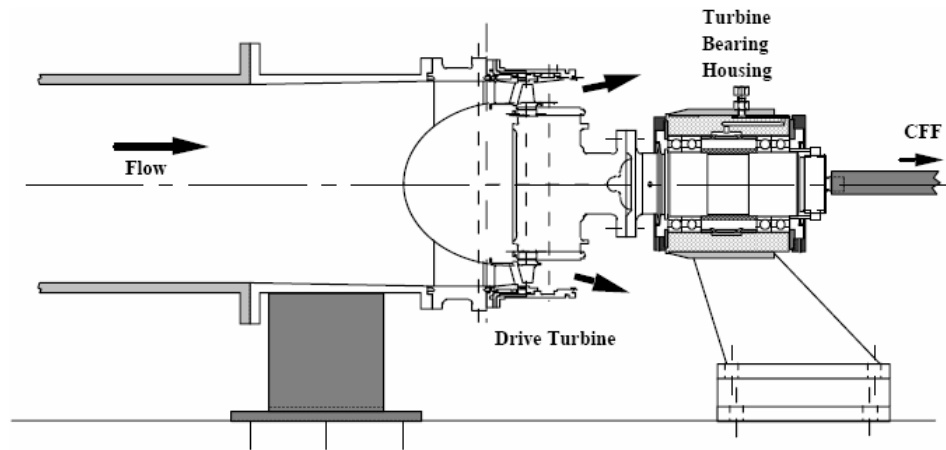
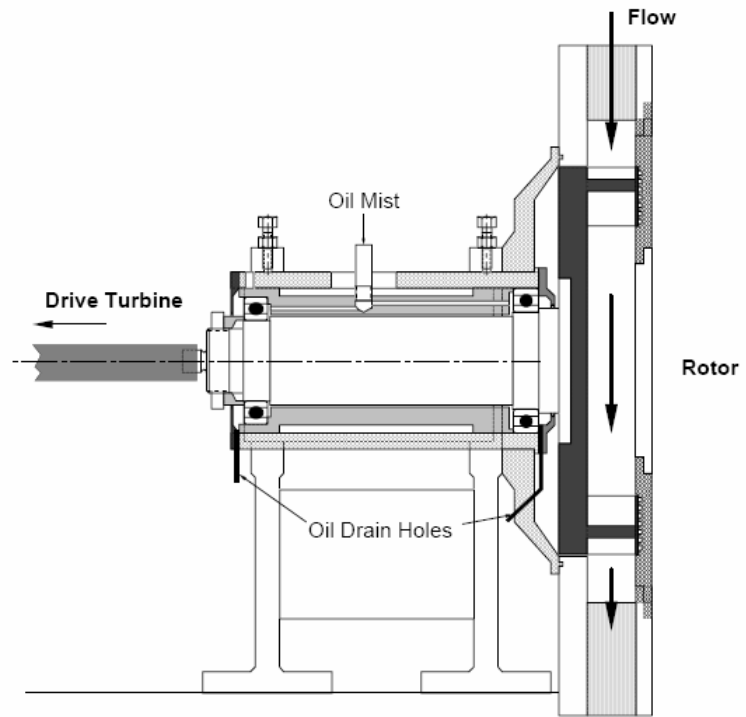


Figure 1. Schematic of the Turbine Test Rig (from Ref 5)

#### 2. Cross-Flow Fan Test Assembly (CFFTA)

The Cross-Flow Fan Test Assembly (CFFTA) design was derived from the VSD Multi-Bypass Ratio System test assembly #6 [Ref. 1]. A schematic of it is shown in Figure 2. The configuration of the CFFTA was composed of a fan rotor (with 30 blades, 12-inch diameter and 1.5 inch span), two cavity components, an exhaust duct wall, a drive shaft, and bearing housing.

For the initial tests, unlike [Ref. 3] and [Ref. 4], an inlet bell-mouth was used to measure the mass flow rate, which was denoted Test Plan I. For the second test, an extended exhaust duct and butterfly throttle valve, were added in order to measure the performance at various mass-flow rates. This was denoted as Test Plan II.



(a) Schematic



(b) Fan Rotor



(c) Partially Assembled  
Figure 2. Cross-Flow Test Assembly (CFFTA, from Ref. 5)

### 3. Control Station

The CFFTA was controlled by adjusting a bypass dump valve, and an inlet valve to the TTR. Atmospheric pressure, rotational speed of the fan, bearing temperatures and vibrations were monitored, and were recorded in a logbook during the test, once a desired rotational speed and throttle setting was stably achieved

### 4. Data Acquisition and Instrumentation

Instruments employed to measure the flow properties were United Sensor Devices model USD-C-161 3 mm (1/8-inch) combination thermocouple/pressure probes, and static pressure taps. The locations of the probes, which are shown in the schematic in Figure 3, were slightly different from [Ref 3] and [Ref 4]. Two more probes were added in the exhaust duct, and one was placed in the inlet section at the 12 o'clock position. The type of measurement and labels of probes are listed in Table 1 and Table 2.

It was concluded that the variation of total pressure of the three inlet combination probes was not significant; therefore, they were averaged by introducing "tees" to combine them before being connected to the pressure transducer array. In the same manner, this averaging technique was applied to the three static pressure tapes at the throat of the bell-mouth and to the two exhaust duct static pressure taps.

Once the data were acquired, the pressures were converted from analog signals through the digital sensor array (DSA) into digital signals which were acquired by the

computer via an internet cable. Temperatures were recorded by a Hewlett- Packard HP E1326B Multimeter Adapter, which was connected to computer via the VXI mainframe.

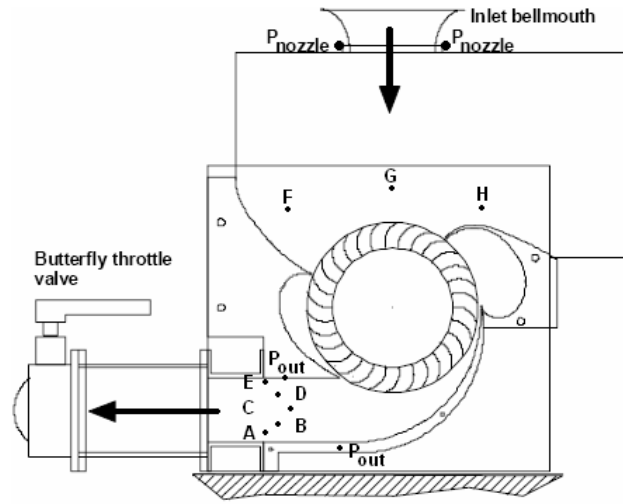


Figure 3. Combination probes and pressure tap layout

Multiplexer Channel	Probe	Nomenclature
4	A	$T_{t,out,A}$
13	B	$T_{t,out,B}$
14	C	$T_{t,out,C}$
15	D	$T_{t,out,D}$
5	E	$T_{t,out,E}$
6	F	$T_{t,in,F}$
8	G	$T_{t,in,G}$
9	H	$T_{t,in,H}$
10		$T_{t,in,TTR}$
11		$T_{t,out,TTR}$
12		$T_{t,in,orifice}$

Table 1. Thermocouple scanning multiplexer channel assignments

Port #	Probe	Nomenclature	Type
4	A	$P_{t,out,A}$	Total Pressure
5	B	$P_{t,out,B}$	Total Pressure
6	C	$P_{t,out,C}$	Total Pressure
7	D	$P_{t,out,D}$	Total Pressure
8	E	$P_{t,out,E}$	Total Pressure
3	F	$\bar{P}_{t,in}$	Total Pressure
	G		
	H		
2		$P_{nozzle}$	Static Pressure
1		$\bar{P}_{out}$ (for Test Plan II only)	Static Pressure

Table 2. Scanivalve port assignments

## B. DATA ACQUISITION AND REDUCTION

The software Agilent VEE, which is a window-based data acquisition program, was used to develop a new graphical use interface (GUI), called CFF\_DAQ, to acquire all temperatures, pressures and the rotational speed of the CFF. This is shown in Figure 4. All data were written to a text file, which EXCEL could read, and then data reduction procedures were performed to calculate parameters of interest. The program structure of CFF\_DAQ is given in APPENDIX A.

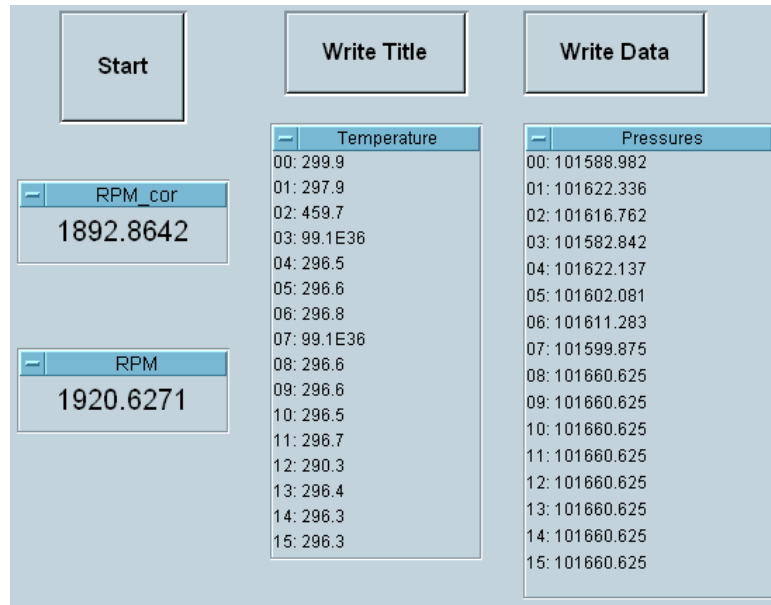


Figure 4. CFF\_DAQ graphical user interface

During the post processing, several properties had to be related since mass-flow rate, total pressure ratio, total temperature ratio, efficiency, thrust, and power had to be calculated. Data reduction included referring to standard day conditions ( $T_{t,std} = 288.15K$  and  $P_{t,std} = 101,325Pa$  ).

The mass flow rate was obtained from the bell-mouth measurements, and the basic equations used for the calculation are presented below. For a detailed derivation, please refer to APPENDIX B.

$$\dot{m}_{in} = \frac{P_1}{RT_1} AV_{in} \quad (1)$$

$$T_1 = T_{t1} - \frac{(\gamma - 1)}{2\gamma R} V_{in}^2 \quad (2)$$

$$V_{in} = \sqrt{\frac{2RT_{t1}}{\frac{1}{\gamma} + \left(\frac{\gamma - 1}{2}\right)\left(\frac{P_{t1}}{P_1}\right)}} \left(\frac{P_{t1}}{P_1} - 1\right) \quad (3)$$

where  $P_1 = P_{nozzle}$  ;  $A$  is the cross-area of the bell-mouth;  $P_{t1} = P_{atm}$  , which was read from the control station;  $T_{t1} = T_{t,in}$  , which was an average of  $T_{t,in,F}$  ,  $T_{t,in,G}$  and  $T_{t,in,H}$  . (See Figure 3)

Since there were five probes placed in the outlet section, as shown in Figure 5, five zones (from A to E) were utilized to calculate mass-averaged outlet properties of total pressure and temperature. The expressions of compressible flow were introduced from [Ref. 7] and used to evaluate the mass flow rate within each zone;

$$\dot{m}_i = X_i \left(1 - X_i^2\right)^{\frac{1}{\gamma-1}} \frac{P_{t,i}}{RT_{t,i}} \sqrt{2c_p T_{t,i}} A_i \quad (4)$$

where  $X$  was defined as a fraction equal to the velocity referred to the stagnation velocity. This was calculated from the following relationship between local static and stagnation pressures;



$$X_i = \sqrt{1 - \left( \frac{P}{P_{t,i}} \right)^{\frac{\gamma-1}{\gamma}}} \quad (5)$$

where

$$P = P_{atm} \text{ (for Test Plan I), } P = \bar{P}_{t,out} \text{ (for Test Plan II),}$$

and  $i$  indicates each zone.  $R = 287 \text{ m}^2 / \text{s}^2 \cdot \text{K}$ ;  $c_p = 1004.4 \text{ J} / \text{kg} \cdot \text{K}$ ;  $\gamma = 1.402$ ;  $P_{t,i}$  and  $T_{t,i}$  were measured from the five combination probes;  $A_i$  was the area of each zone and the values provided in Table 3 were measured according to the actual locations of the probes.

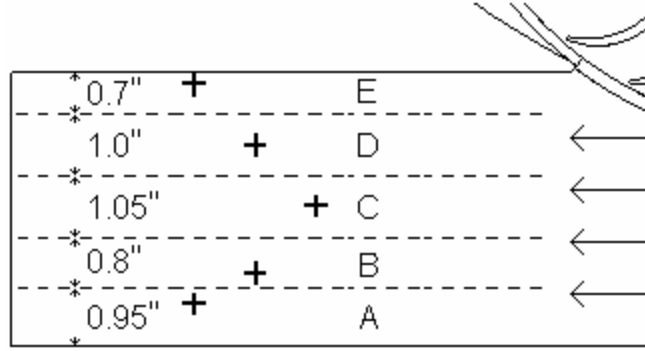


Figure 5. Schematic of the position of the combination probes in the exhaust duct

Zone	Area [ $\text{m}^2$ ]
A	0.000919353
B	0.000774192
C	0.001016127
D	0.00096774
E	0.000677418

Table 3. Area values of each zone

Thus, the mass averaged total pressure and temperature in the exhaust duct were obtained from;

$$\bar{P}_{t,out} = \frac{\dot{m}_A P_{t,out,A} + \dot{m}_B P_{t,out,B} + \dot{m}_C P_{t,out,C} + \dot{m}_D P_{t,out,D} + \dot{m}_E P_{t,out,E}}{\sum_{i=A}^E \dot{m}_i} \quad (6)$$

$$\bar{T}_{t,out} = \frac{\dot{m}_A T_{t,out,A} + \dot{m}_B T_{t,out,B} + \dot{m}_C T_{t,out,C} + \dot{m}_D T_{t,out,D} + \dot{m}_E T_{t,out,E}}{\sum_{i=A}^E \dot{m}_i} \quad (7)$$

The total pressure ratio ( $\pi$ ), total temperature ratio ( $\tau$ ) and efficiency ( $\eta$ ) were then found by;

$$\pi = \frac{\bar{P}_{t,out}}{\bar{P}_{t,in}} \quad (8)$$

$$\tau = \frac{\bar{T}_{t,out}}{\bar{T}_{t,in}} \quad (9)$$

$$\eta = \frac{\pi^{\frac{\gamma-1}{\gamma}} - 1}{\tau - 1} \quad (10)$$

Next, the thrust force was obtained from;

$$F_{thrust} = \dot{m}_{in} (u_{out} - u_{in}) \quad (11)$$

where  $u_{in}$  was assumed to be zero and;

$$u_{out} = M_{out} \sqrt{\gamma R T_{out}} \quad (12)$$

$$T_{out} = \frac{\bar{T}_{t,out}}{1 + \frac{\gamma-1}{2} M_{out}^2} \quad (13)$$

$$M_{out} = \left\{ \frac{2}{\gamma-1} \left[ \left( \frac{\bar{P}_{t,out}}{\bar{P}_{atm}} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right] \right\}^{\frac{1}{2}} \quad (14)$$

Finally, the power absorbed by the CFF was calculated from;

$$P_{ower} = \dot{m}_{in} c_p (\bar{T}_{t,out} - \bar{T}_{t,in}) \quad (15)$$

After all the parameters were calculated, the correction to standard day conditions was accomplished as follows;

$$\delta = \frac{P_{t,in}}{P_{t,std}}; \theta = \frac{T_{t,in}}{T_{t,std}} \quad (16)$$

$$\dot{m}_{corr} = \dot{m} \frac{\sqrt{\theta}}{\delta}; N_{corr} = \frac{N}{\sqrt{\theta}}; F_{corr} = \frac{F}{\delta}; P_{corr} = \frac{P}{\delta \sqrt{\theta}} \quad (17)$$

### C. TEST PLANS

Two test plans were performed with both cavities opened; these were, Test Plan I - the Baseline Configuration, which was run from 1000RPM up to 5000RPM, and Test Plan II - the Throttling Investigation, where throttle settings were employed to control the mass flow rate at particular rotational speeds. The complete assemblies of both configurations are shown in Figure 6.



(a)



(b)  
Figure 6. CFFTA for (a) Test Plan I and (b) Test Plan II

### **III. DESCRIPTION OF THE NUMERICAL MODEL**

#### **A. OVERVIEW**

In order to study characteristics of a flow in a cross-flow fan, the commercial computational fluid dynamics (CFD) package, CFX-5, which was developed by ANSYS Inc, was used to more accurately model the compressible flow through the CFF. Three modules were included; CFX-Pre, CFX-Solver and CFX-Post. In CFX-Pre, a simulation type was defined; rotating or stationary domain, boundary conditions and the interface surfaces were defined; initial values, solver controls and output controls were set. After initializing the information required by CFX-Pre, the simulation was started by writing files to the CFX-Solver. Convergence histories were monitored while running with the solver, which also wrote out result files (\*.res and \*.out). By using CFX-Post, the results could be read and plotted for visualization and further post processing.

#### **B. GEOMETRY AND GRID GENERATION**

The geometry for the 12 inch diameter rotor with 30 blades, based on the experiment by Seaton and Cheng [Ref 5 & 1], was created by using ICEM-CFD, which was compatible with CFX-5. Since a finite-volume approach was utilized in CFX-5, a 3-dimensional geometry was required and generated with a span of 0.1 inch. It was then treated as a 2-D case, due to there being only one grid point in the span-wise direction.

There were ten volume meshes generated separately throughout the geometry. Two were in the rotational domain, which were identified as BLADES and INNERCIRCLE, and eight parts were in the stationary domain, which were identified as INLET, OUTLET, High Pressure Cavity (HPC), Low Pressure Cavity (LPC), INLET(Interface), OUTLET(Interface), HPC(Interface) and LPC(Interface). Attention needed to be paid to the last four parts in the stationary domain, since they were adjacent to the rotational domain as shown in Figure 7. A complete volume mesh was created by merging each part together, which was done by opening individual mesh files (\*.uns) sequentially. Therefore, 2D planes of each part sharing a common boundary between the interfaces were generated, which are shown in Figure 8.

It was believed that the shape of the blades influenced the performance of the cross-flow fan. Thus O-grid topologies were employed for the regions around the blades in order to more accurately model the blade surface boundary layers, as shown in Figure 9. Hexahedral cells were generated for all domains. A procedure for generating this geometry and mesh are provided in APPENDIX D.

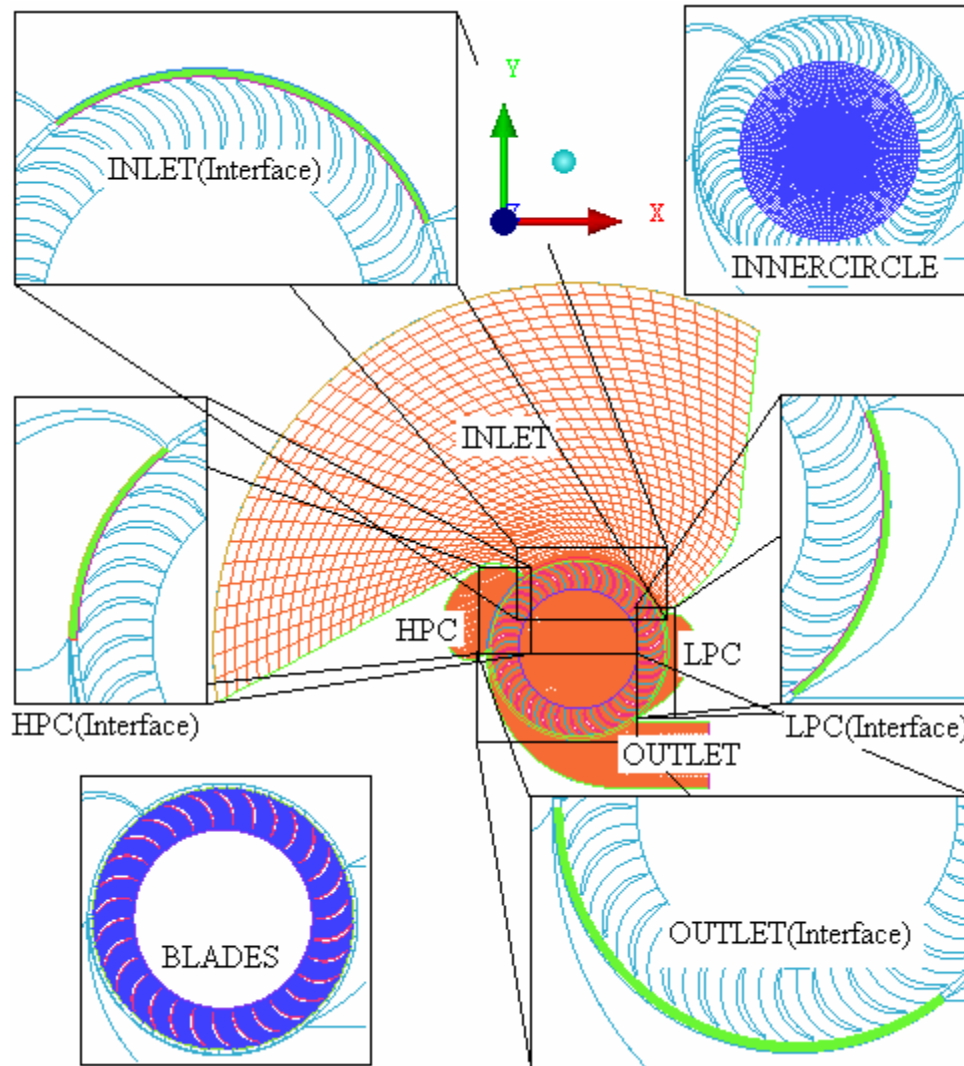


Figure 7. Identification of the ten domains of the CFF mesh

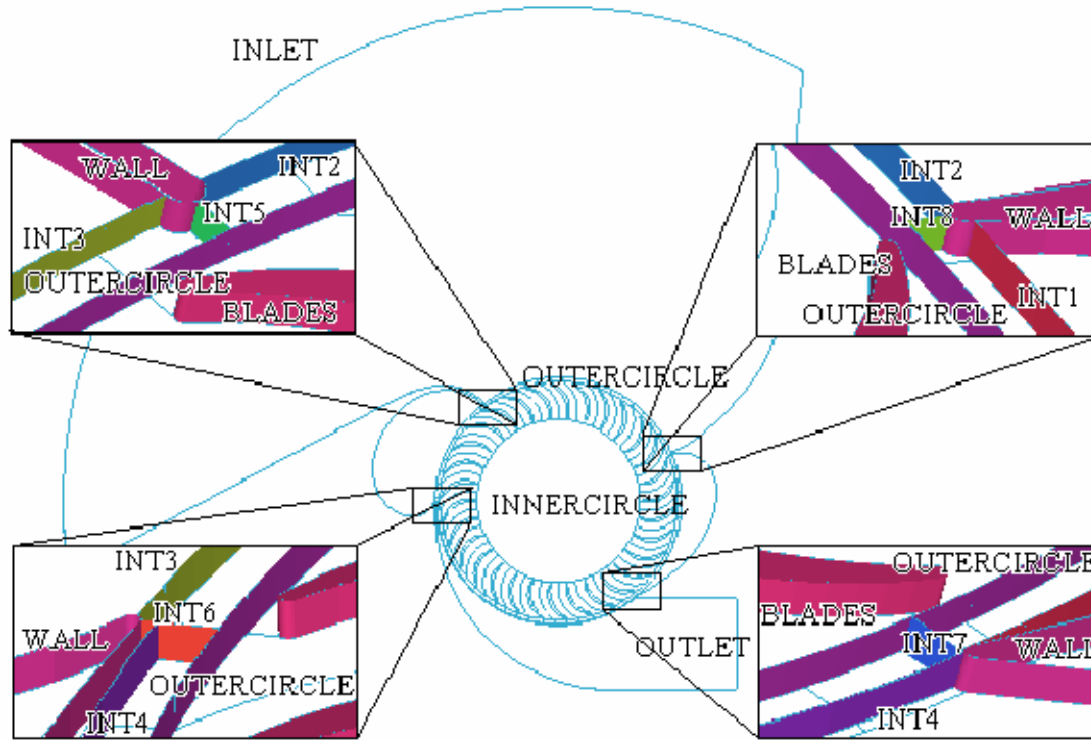


Figure 8. The planes between the interfaces

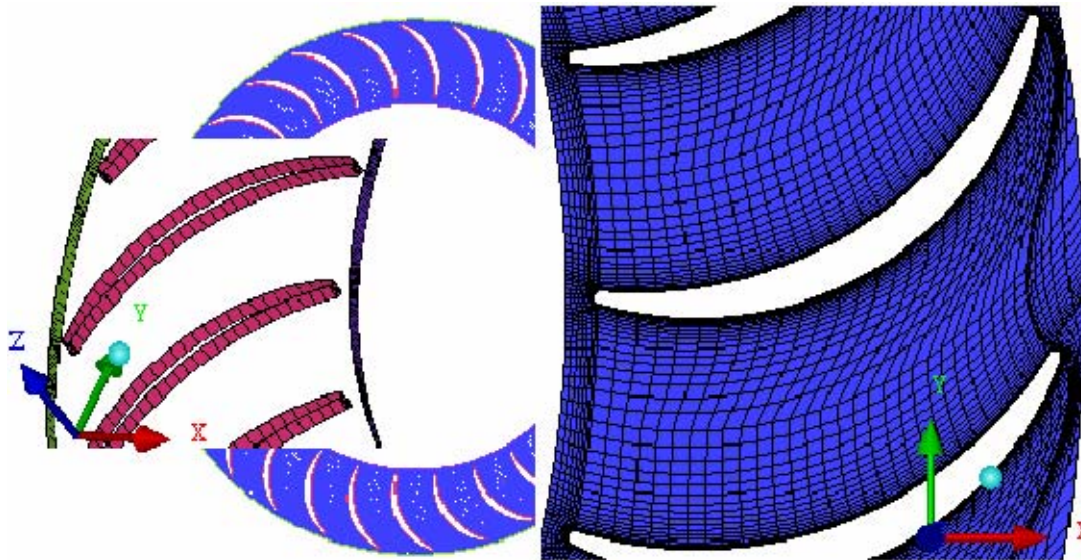


Figure 9. The meshes around blades

### C. METHODOLOGY AND BOUNDARY CONDITIONS

Transient solutions were carried out by setting the interface between the rotational domains and the stationary domains within the model to “Transient Rotor-Stator”, and the sliding interfaces were then specified. The fluid was specified as an ideal gas with

constant specific heat at constant pressure. With a reference pressure of 1 atm, the opening inlet and opening outlet boundary conditions were set to an average relative total pressure of 0 atm, and a relative static pressure of 0 atm, respectively. The reason for using an opening boundary condition, at both the inlet and exit, was because of uncertainty in flow recirculation at both boundaries. Static temperature of 300K, or standard day temperature, and turbulent intensity of 5% were also set for both inlet and outlet boundaries. An adiabatic smooth wall boundary with no slip was imposed, throughout on all walls.

Total energy and the k-epsilon turbulence model were used to include effects of heat transfer and turbulence, respectively. By setting a transient initialisation, the simulation could be run without initial values interpolated; the solver default values were employed at the beginning of the run. Equations involved in the solver were continuity, momentum, energy, turbulence eddy dissipation, turbulence kinetic energy, and an equation of state. The set-up procedure in CFX-Pre is also provided in APPENDIX E.

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0$$

Momentum equation:

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) = \nabla \cdot \left( -p \delta + \mu (\nabla U + (\nabla U)^T) \right) + S_M$$

Energy equation:

$$\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho U h_{tot}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot \left( \mu \nabla U + \nabla U^T - \frac{2}{3} \nabla \cdot U \delta U \right) + S_E$$

$$h_{tot}(p, T) = h_{stat}(p, T) + \frac{1}{2} U^2$$

Turbulent eddy viscosity

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}$$

Turbulent kinetic energy



$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho U k) = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + P_k - \rho \varepsilon$$

Turbulence Eddy Dissipation

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho U \varepsilon) = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon)$$

The equation of State

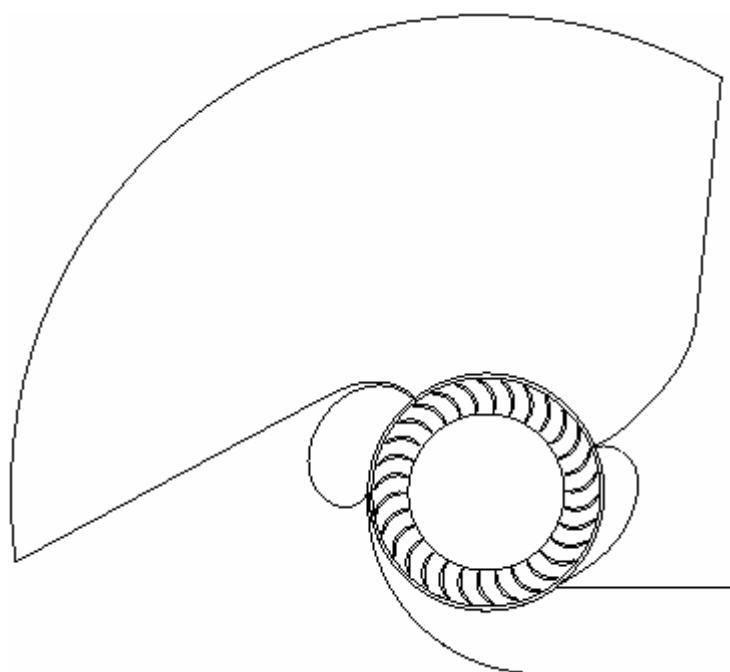
$$\rho(p, T) = \frac{w(p + p_{ref})}{R_0 T}$$

#### D. SIMULATION PLANS

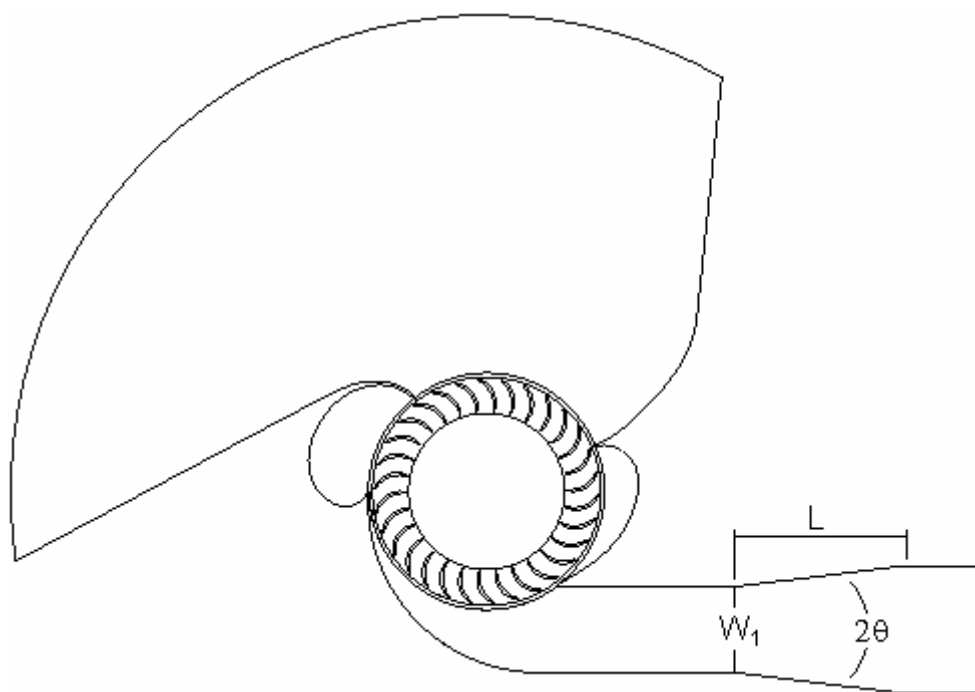
Two simulation plans were embarked on to predict the performance at conditions similar to the experimental test plans. These are summarized in Table 4. The configurations of both plans are presented in Figure 10. As shown, to constrain the inlet flow into the CFF, an extending wall was added instead of the bell-mouth, the mass-flow rate was, therefore, recorded from the inlet boundary for the following analysis. More difficulties were encountered to predict the case of Test Plan II, a suitable diffuser had to be determined to carry out the expected mass flow rate variation since several effects had to be considered, such as inlet turbulence and inlet velocity profile.. From [Ref. 8], the maximum diffuser  $L/W_1 = 2$  and  $2\theta = 14^\circ$  was used. To vary the mass flow rate, the diffuser was closed gradually by 1.75 degree until stall occurred.

Case \ Speed	Simulation Plan I (corresponding to Test Plan I)	Simulation Plan II (corresponding to Test Plan II)
1000RPM	V	
2000RPM	V	V
3000RPM	V	V
4000RPM	V	
5000RPM	V	V

Table 4. Summary of simulation plans



(a)



(b)

Figure 10. Geometric profile of (a) Simulation Plan I and (b) Simulation Plan II

## **IV. RESULTS AND DISCUSSION**

### **A. MESH SENSITIVITY STUDY**

The mesh sensitivity was studied under a rotational speed of 3000RPM with a time-step of  $3.33333\text{e-}5$  seconds, which corresponded to a rotation angle of 0.6 degrees per time step.

At first, with a constant number of mesh elements (62,493), the sensitivity was studied by varying the spacing of grids of specific parts, and run for 10 revolutions. Since meshes were not distributed uniformly along the interfaces, the confirmation of transferring properties between domains was required. It was required that the differences between simulated results were less than 0.06%. The best results were obtained with the most clustering of points close to the interfaces. Two more different mesh sizes were modeled, the coarse one with 13,657 elements and the finest one with 250,573 elements. Once the calculations were completed, the efficiency was used as an indicator to determine the optimum mesh size. It was found that the coarse mesh size provided an efficiency of 57.26%, and the finest one provided an efficiency of 76.16%, those were not close to the expected efficiency of 68%. The only explanation for this was that the errors accumulated in the finest meshes, which adversely effected in the transient model.

However, a medium mesh size provided the efficiency of 68.13%, which corresponded favorably with the experimental value. In addition, its corresponding corrected mass flow rate of 0.4932 kg/s was calculated, which was close to the experimental value of 0.4872 kg/s. Therefore, the mesh size of 62,493 elements was used in the subsequent simulations. All the simulations were for 10 revolutions, the reason to be discussed in the following section on convergence history.

### **B. TIME-STEP SELECTION**

Generally, smaller time-steps need more time to calculate but provide more accurate results. The goal of the present study was to find the optimal time-step, the one that would not only take less time for the simulation, but also provide an acceptable solution. Therefore, the selection of time-step was performed based on the consequence

of the mesh sensitivity study with a rotating speed of 3000RPM. A computer with Pentium@ 4 CPU with double 3.2Ghz processors and 3.25 GB RAM was used. Four cases were carried out, which were 12 degrees, 6 degrees, 1 degree and 0.6 degrees rotational angle per time step, and all cases were run for only one revolution.

It was found that all residuals (momentum and mass, turbulence, and heat transfer) were converged and fluctuated along  $10^{-3}$ ,  $10^{-3} \sim 10^{-4}$ , and  $10^{-4}$  for the first three cases, and computational times were 1 hours, 2 hours and 13 hours, respectively. For the last case, the residuals were just slightly lower than  $10^{-4}$  but required 30 hours. Hence, 1 degree angle per time step was selected for further simulations.

## **C. INITIAL SIMULATION RESULTS**

A baseline model at 3000RPM was first simulated for 10 revolutions to see if the converged solutions could be calculated. To compare with the results of experiments, the same locations of probes were used in the computational model. For each revolution, the parameters, such as mass-flow rate, total pressure ratio, total temperature ratio and efficiency, were calculated and monitored in order to check whether steady solutions were achieved.

### **1. Convergence History**

The calculated parameters which were used for monitoring purposes were slightly different from those used for comparison to the experimental results. Since the expression of compressible flow (Eq. 5) may result in imaginary numbers, the mass flow rate in each zone within the exhaust duct was evaluated directly from the product of density and velocity read from probes in the model. The thrust was derived in the same manner.

The convergence histories for the baseline test case at 3000RPM are shown in Figures 11-13. Figure 11 indicates that the simulation was started from no initialized flow field and it almost reached a stable value after 5 revolutions. This was also observed for the total pressure in Figure 12, however, the total temperature ratio still increased slightly, which affected the efficiency variation until the 8<sup>th</sup> revolution.

The variation of efficiency for the last three revolutions in each model was about 0.004, therefore the solution at the 10th revolution was considered to be stable. All

subsequent calculations were therefore carried out until 10 revolutions had been completed. The convergence histories of all subsequent models were monitored in the same manner.

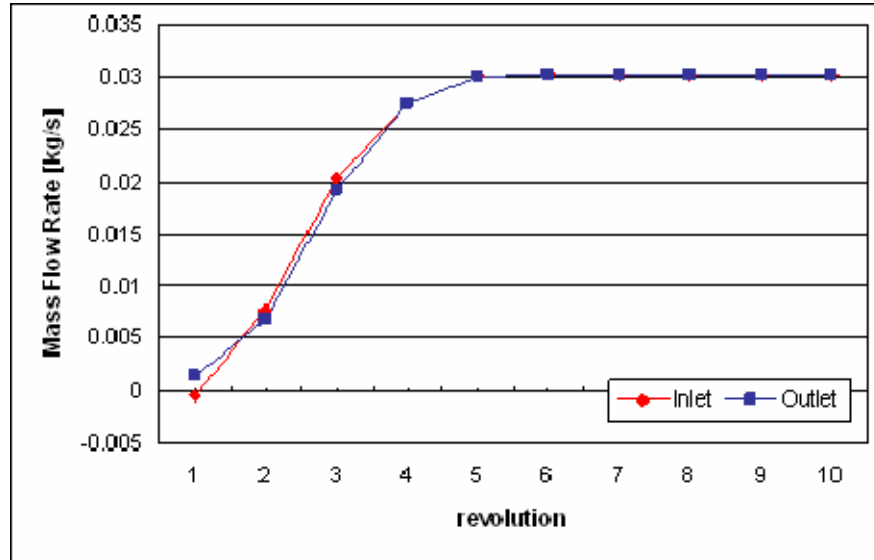


Figure 11. Convergence history of mass flow rate at 3000RPM

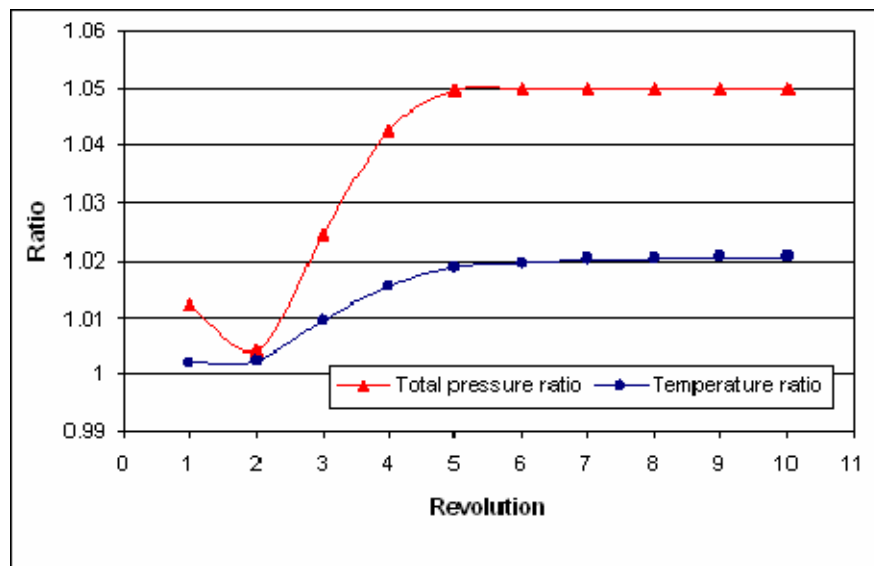


Figure 12. Convergence history of total pressure ratio and total temperature ratio at 3000RPM

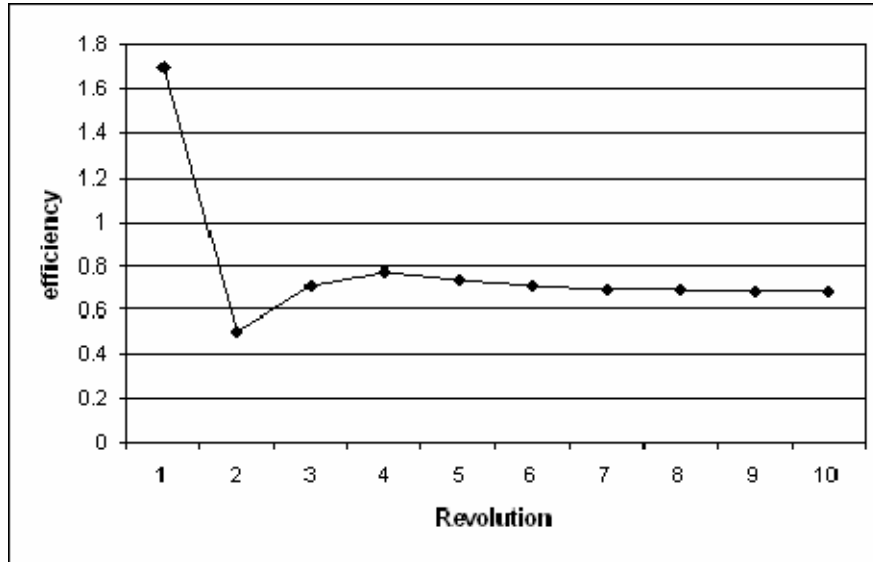


Figure 13. Convergency history of efficiency at 3000RPM

## 2. Flow Visualization

Figure 14 shows vector distributions of flow velocity for both cavities. In the high pressure cavity (HPC), the direction of flow was clockwise, but counterclockwise in the low pressure cavity (LPC), which agreed with the observation in [Ref. 4]. Additional vortical flow features occurred between the blades. The highest velocity region was found within the LPC, consistent with the low pressure. Similarly, the lowest velocity region was found within the HPC, consistent with the high pressure, which explained the names of these cavities.

Figure 15 shows that the lowest pressure did not occur within the LPC, but rather within the rotor close to the LPC. The total pressure profile at the outlet boundary suggested more measuring probes should be used to provide more accurate experimental results.

Figure 16 indicates that the most energy lost was due to the vortex that occurred around the LPC. More heat was generated within this region, which caused the temperature at the LPC to be higher than at the HPC. The exit plane total temperature distribution was non-uniform, with the highest temperature close to the top wall; however the variation of temperature was less than 10 degrees.,

Figure 17 shows the distribution of Mach number, which supports the assumption of subsonic flow within the CFF at 3,000 RPM. A large recirculation in the HPC was observed. The highest Mach numbers ( $\sim 0.39$ ) occurred at three locations, one was within the blades near to the LPC; a second one was in the LPC, and the third one was near to where the flow entered the exhaust duct. The velocity profile at the exhaust duct exit plane is also shown. It was observed that the flow entrainment into the CFF was roughly at the one o'clock position into the rotor..

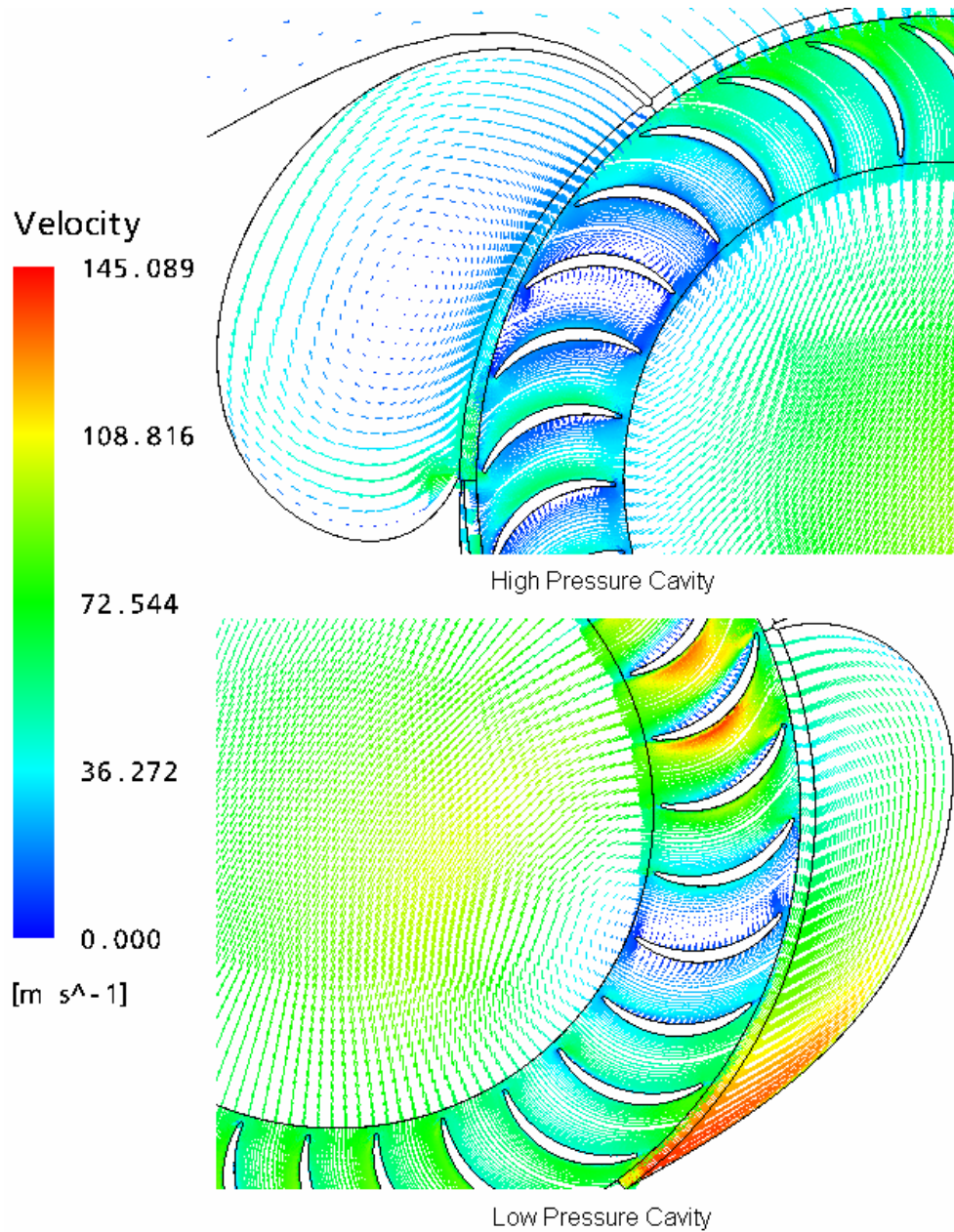
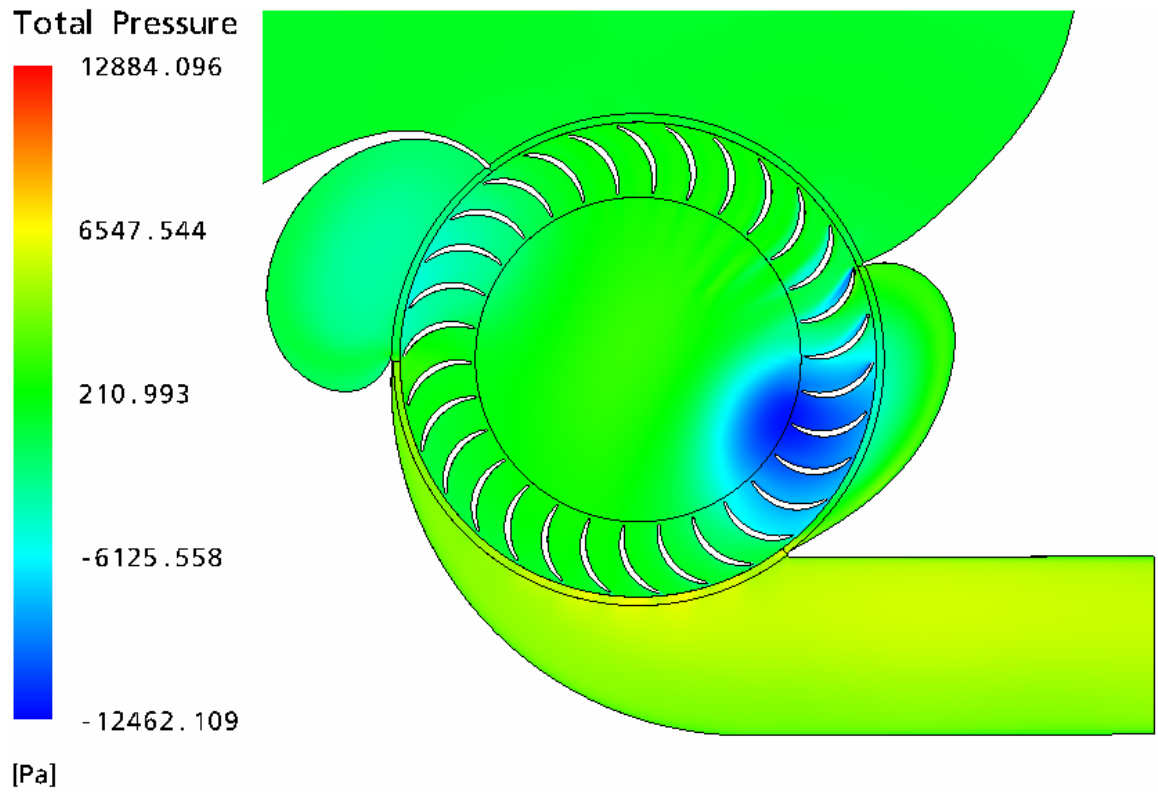
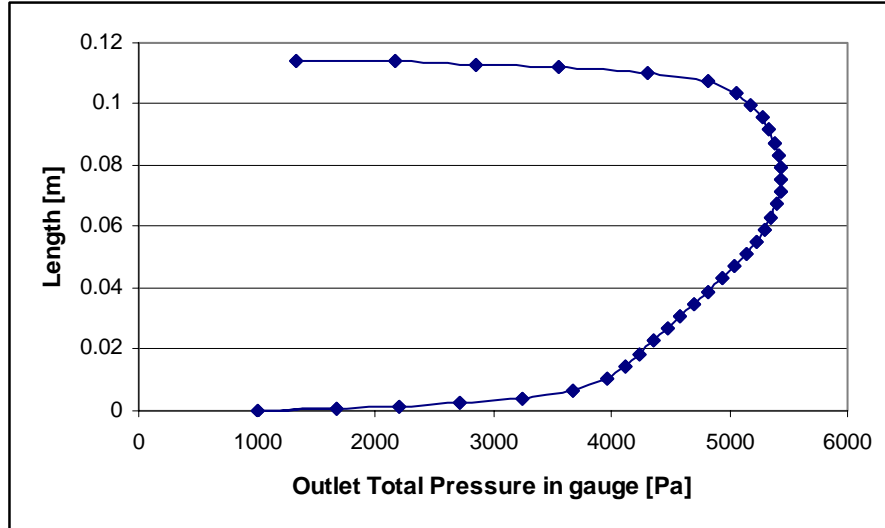


Figure 14. Vector plot of velocity in HPC and LPC at 3000RPM



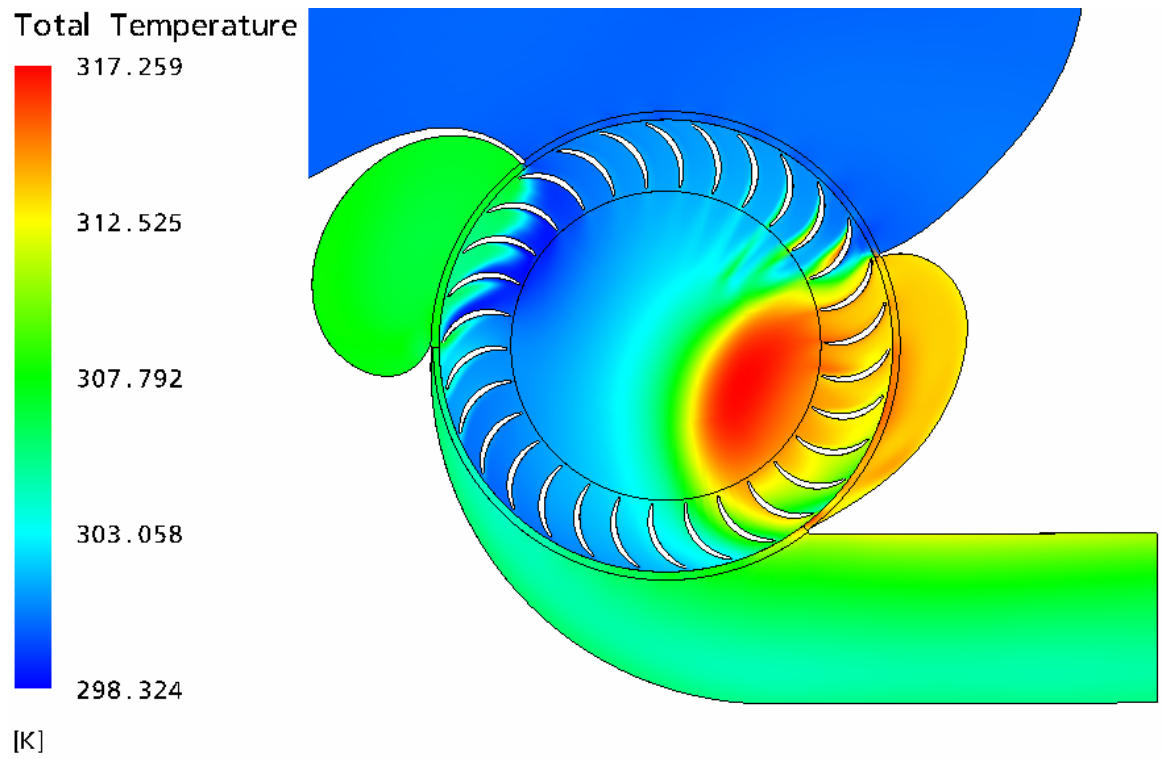
(a) Whole flow field



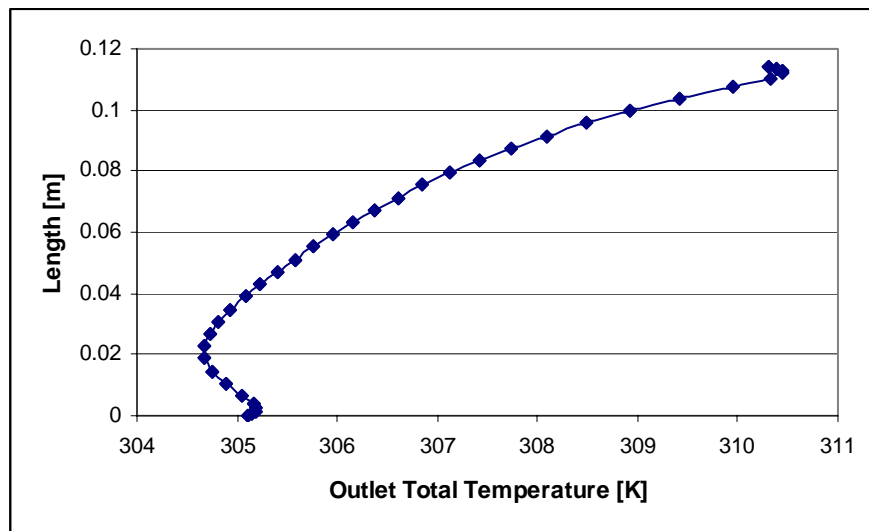
(b) Exit plane

Figure 15. Distribution of total gauge pressure at 3000RPM



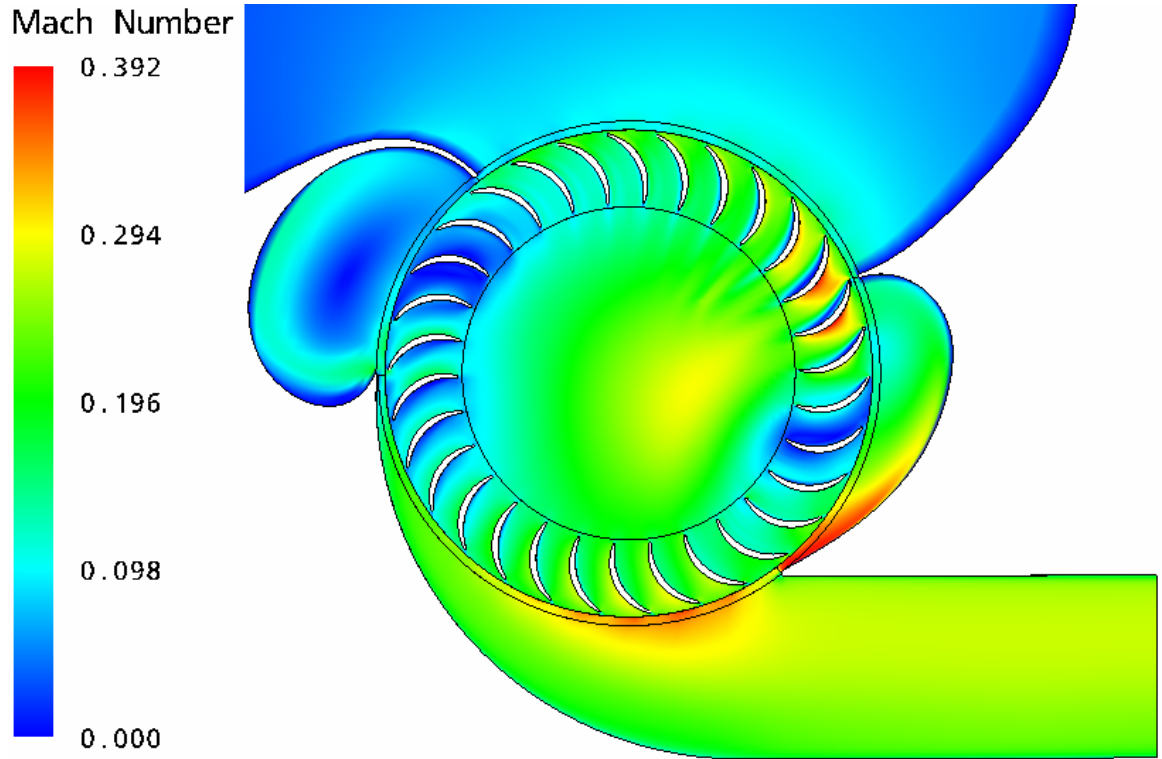


(a) Whole flow field

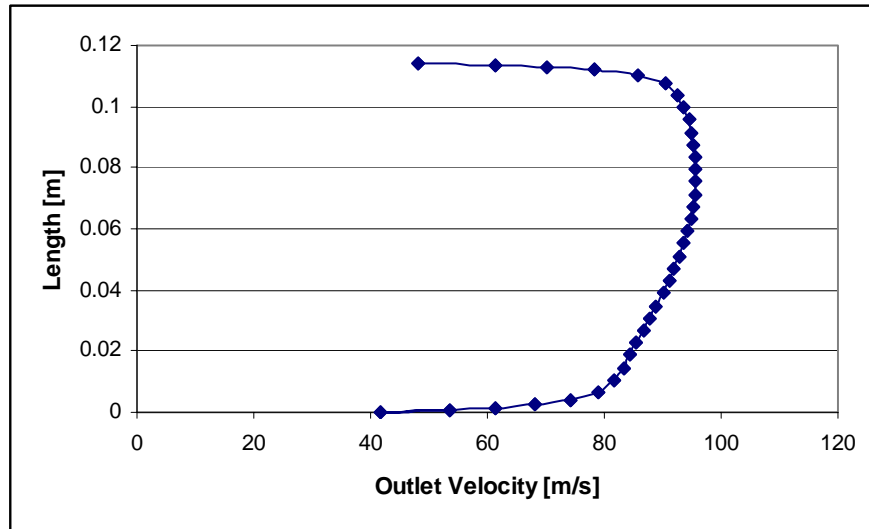


(b) Exit plane

Figure 16. Distribution of total temperature at 3000RPM



(a) Whole flow field



(b) Exit plane

Figure 17. Distribution of Mach number at 3000RPM

#### D. EXPERIMENTAL DATA CONSISTENCY STUDY

The derived parameters from [Ref. 4] were converted from English Engineering (EE) to SI units, and the raw data sets were re-calculated by the procedure mentioned in

Chapter II, section B. Those two data sets were compared to make sure no inconsistencies occurred during the calculations. Since two test plans, with two cavities opened, were tested in this study, only those cases in [Ref. 4] were considered in the following. The results are shown in Figures 18-25.

### 1. Test Plan I

It was expected that the curves re-calculated here [shown as Re-cal] would overlap the existing curves [Ref. 4], in Figures 18-21, since the same procedures were applied. However, it was found that at high rotational speeds, the differences increased, especially in the mass flow rate. However, the differences in the mass flow rates calculated were small, 1.925% maximum, which was considered to be acceptable.

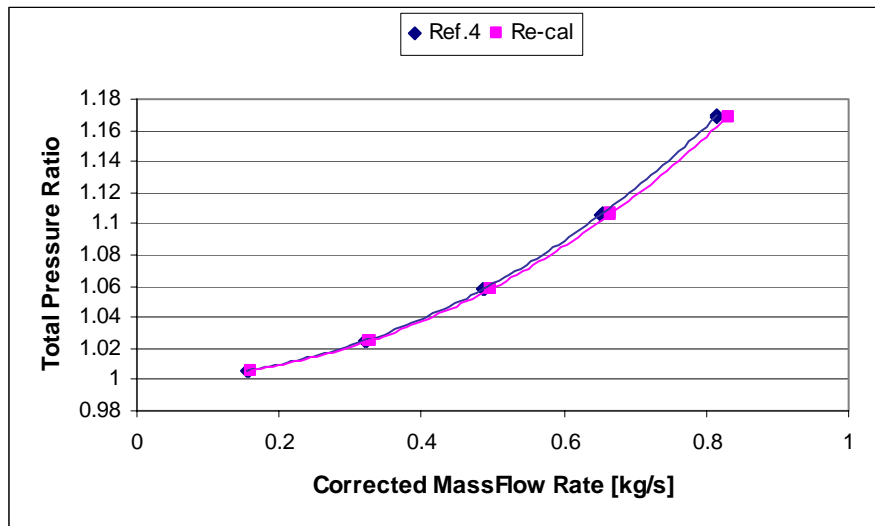


Figure 18. Consistency study of total pressure versus mass flow rate for Test Plan I

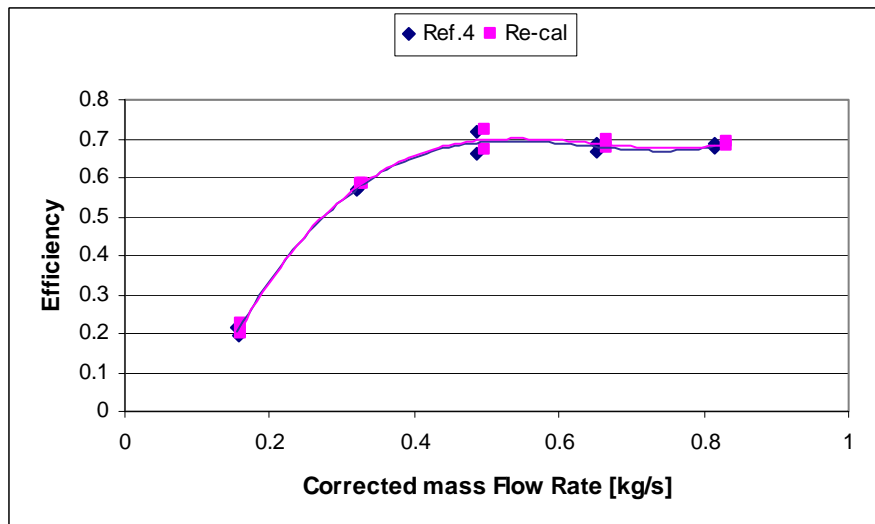


Figure 19. Consistency study of efficiency versus mass flow rate for Test Plan I

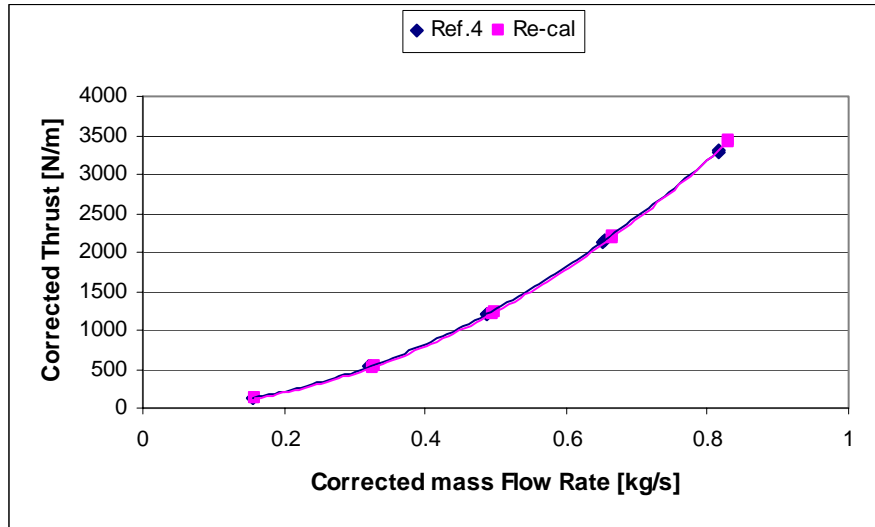


Figure 20. Consistency study of thrust versus mass flow rate for Test Plan I

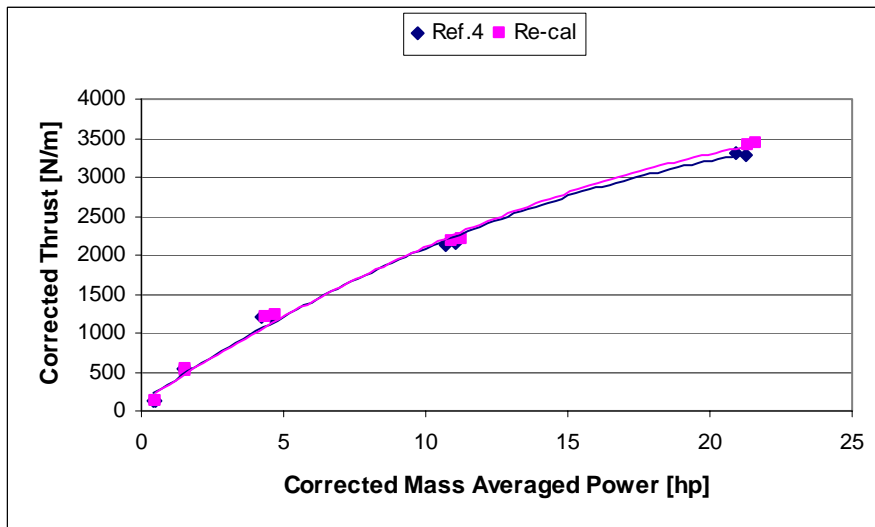


Figure 21. Consistency study of thrust versus power for Test Plan I

## 2. Test Plan II

As in the comparison for Test Plan I, the characteristic curves, shown in Figures 22-25, were not exactly the same, but the trends of total pressure ratio, total temperature ratio and efficiency versus mass flow rate were acceptable since they were close to the values originally calculated in [Ref. 4]. However, the relationship of thrust versus mass flow rate was linear once it was re-calculated; it was not quadratic as presented in [Ref. 4]. And the curves of the power versus mass flow rate shifted up a lot at higher rotational

speeds. Therefore, in order to make the present study consistent, the current calculational procedure was utilized thereafter.

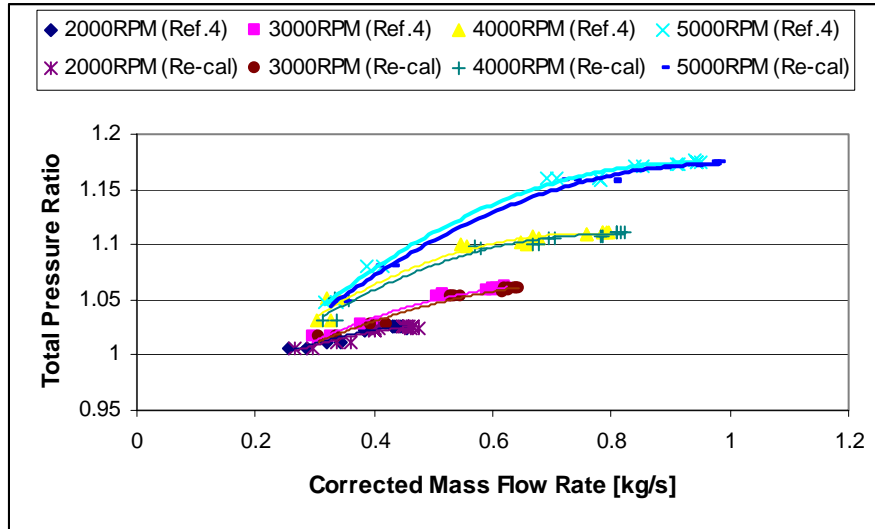


Figure 22. Consistency study of total pressure ratio versus mass flow rate for Test Plan II

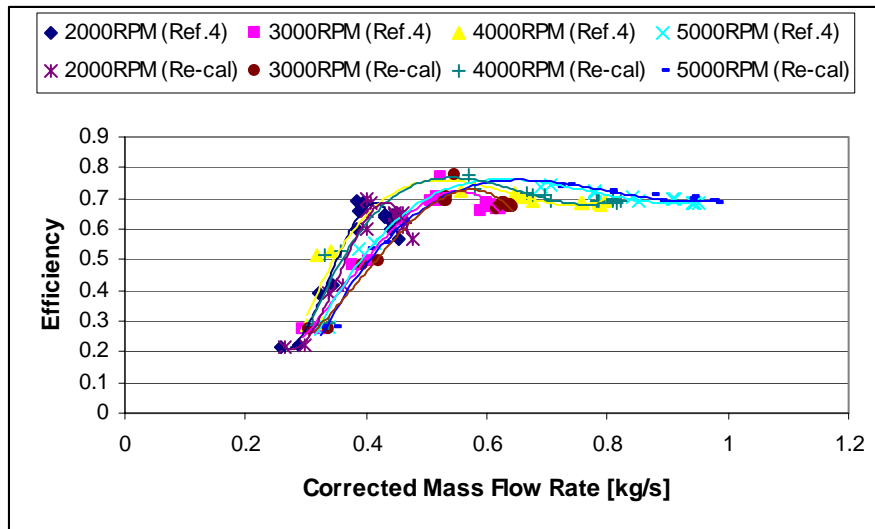


Figure 23. Consistency study of efficiency versus mass flow rate for Test Plan II

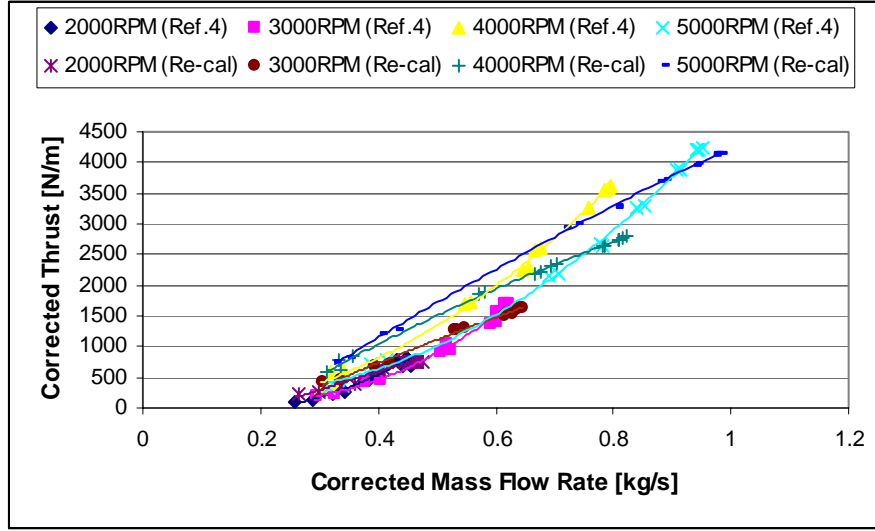


Figure 24. Consistency study of thrust versus mass flow rate for Test Plan II

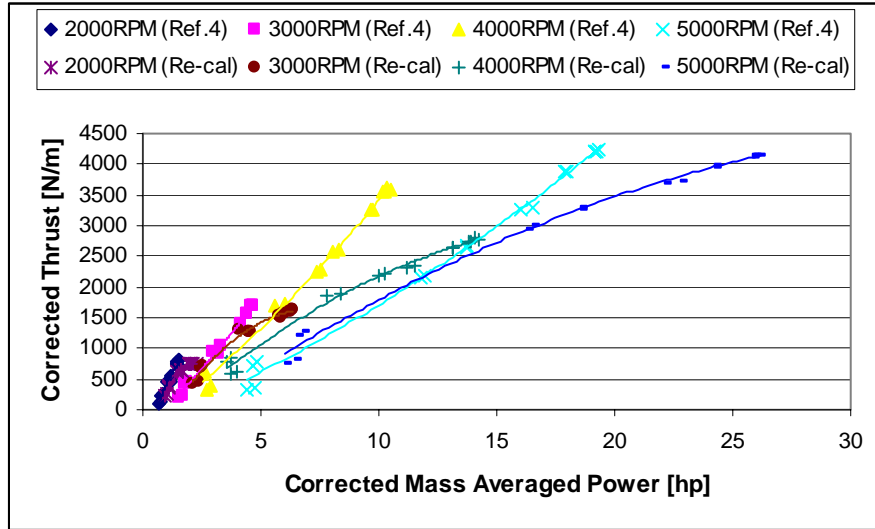


Figure 25. Consistency study of thrust versus power for Test Plan II

#### E. PERFORMANCE OF THE BASELINE CONFIGURATION AT VARIOUS ROTATIONAL SPEEDS

The results of Test Plan I were compared with the results of Simulation Plan I and the results shown in Figures 26-31. The re-calculated results of Cheng [Ref. 4] are also included. The results show a slight difference in the mass flow rate. There was a slight disagreement of the characteristic curves of total temperature ratio, efficiency and power. When the comparison was made of the present experimental data but using different calculational procedures, i.e. using 3 exit combination probes vs 5 probes., more measuring probes improved the agreement in the total pressure ratio and efficiency,

however total temperature ratio and power deviated slightly. The thrust was similar for all calculational procedures and for both data sets, the present as well as [Ref. 4]. The computed (CFD) results were obtained using the same calculational procedure as in the experiment, i.e. using 3 or 5 exit probes. Over the complete speed range the total pressure ratio comparison was very good. However, the curve of the total temperature ratio (with 5 probes) was higher than the experimental, which resulted in an under prediction of the efficiency. At low speeds the computed efficiency was too high, between 60 and 70%, and the trend was not correct as the drop off of efficiency with decreasing speed was not computed.

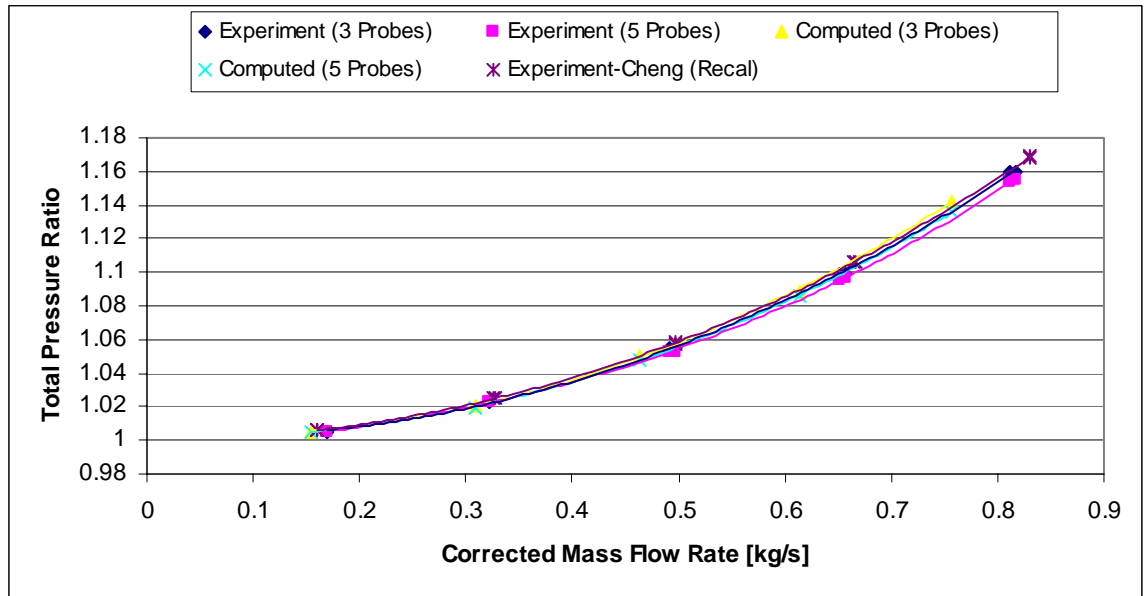


Figure 26. Comparison of total pressure ratio versus mass flow rate for Test Plan I

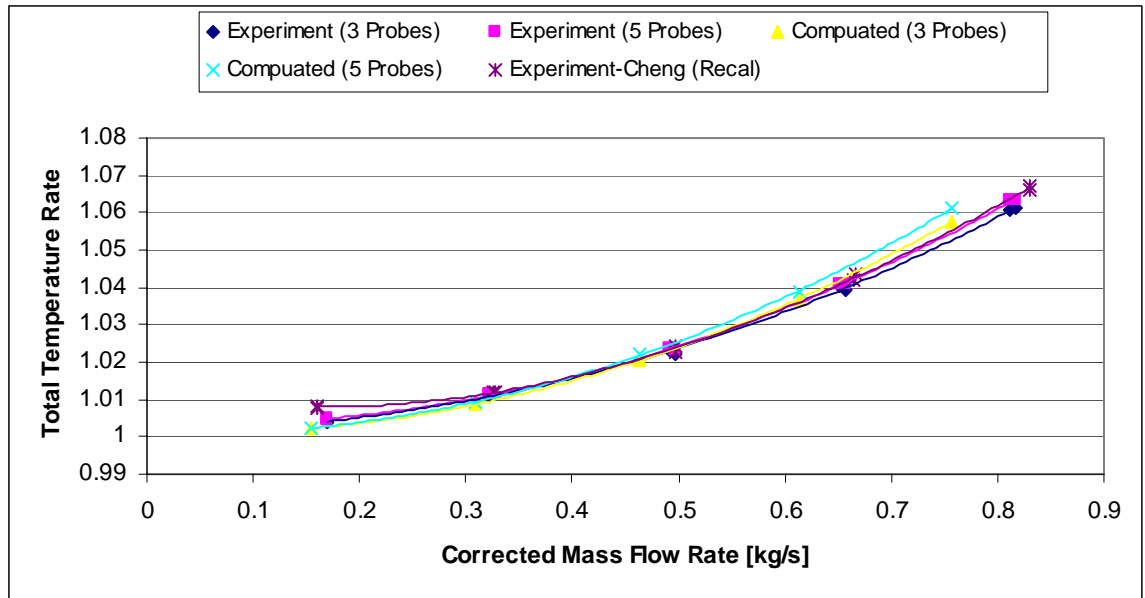


Figure 27. Comparison of total temperature ratio versus mass flow rate for Test Plan I

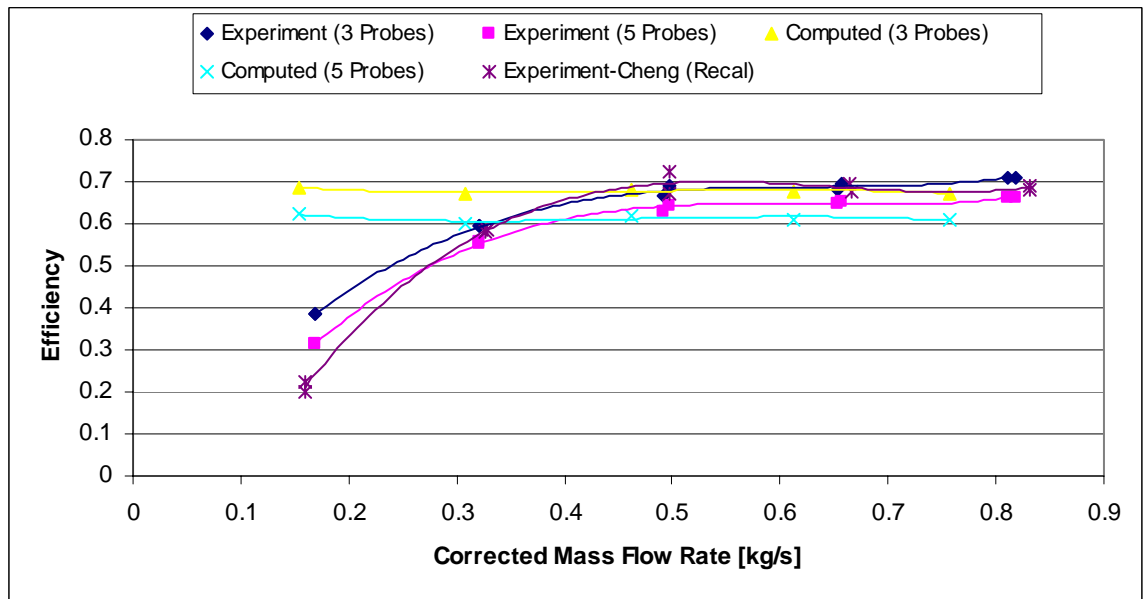


Figure 28. Comparison of efficiency versus mass flow rate for Test Plan I



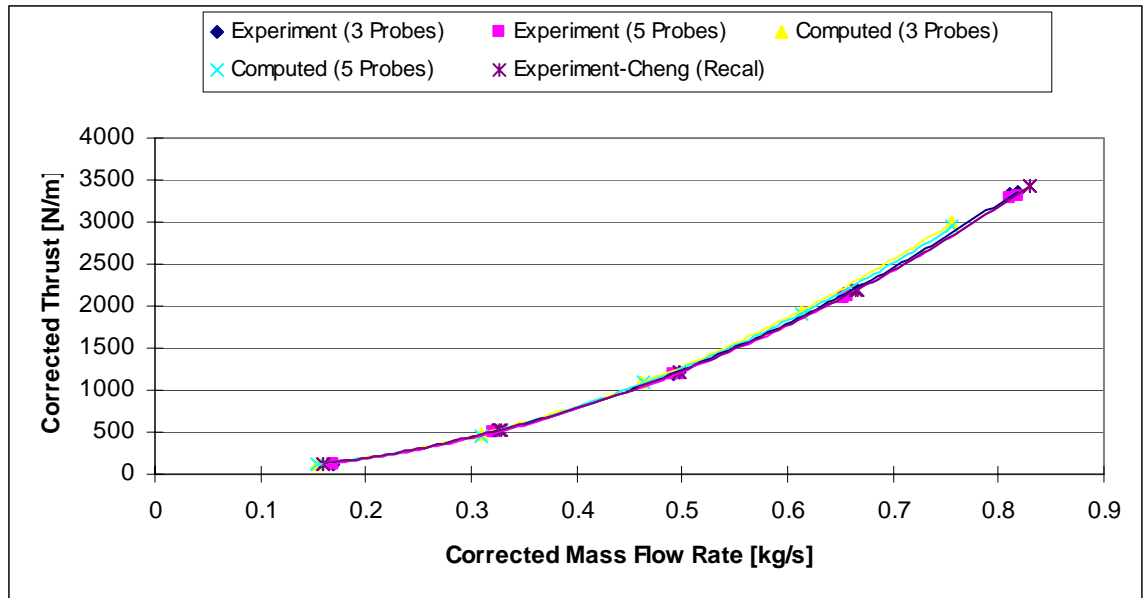


Figure 29. Comparison of thrust versus mass flow rate for Test Plan I

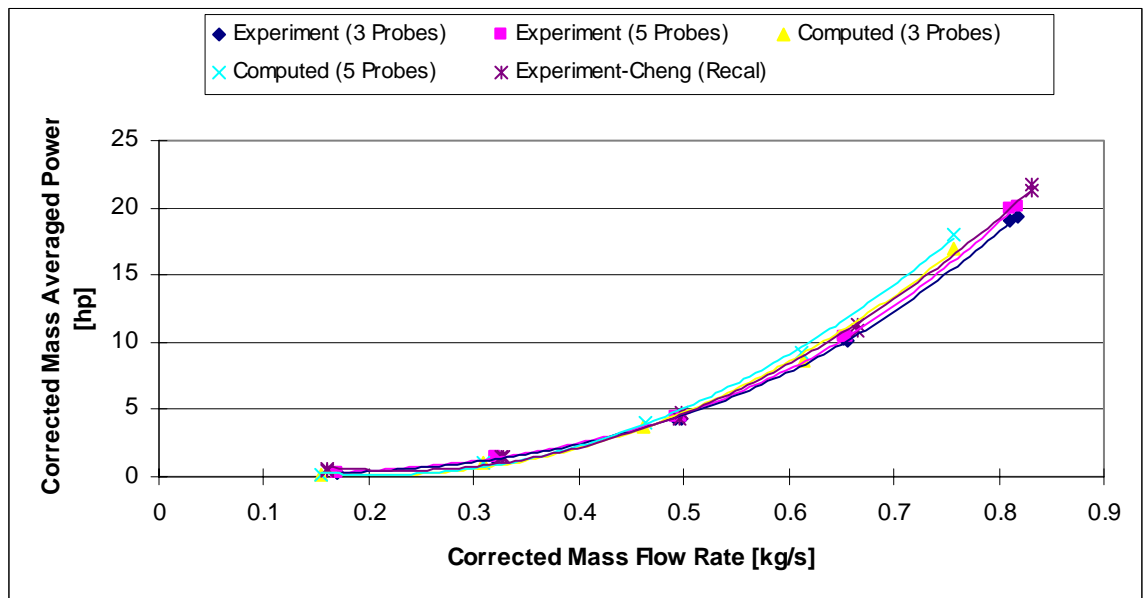


Figure 30. Comparison of power versus mass flow rate for Test Plan I

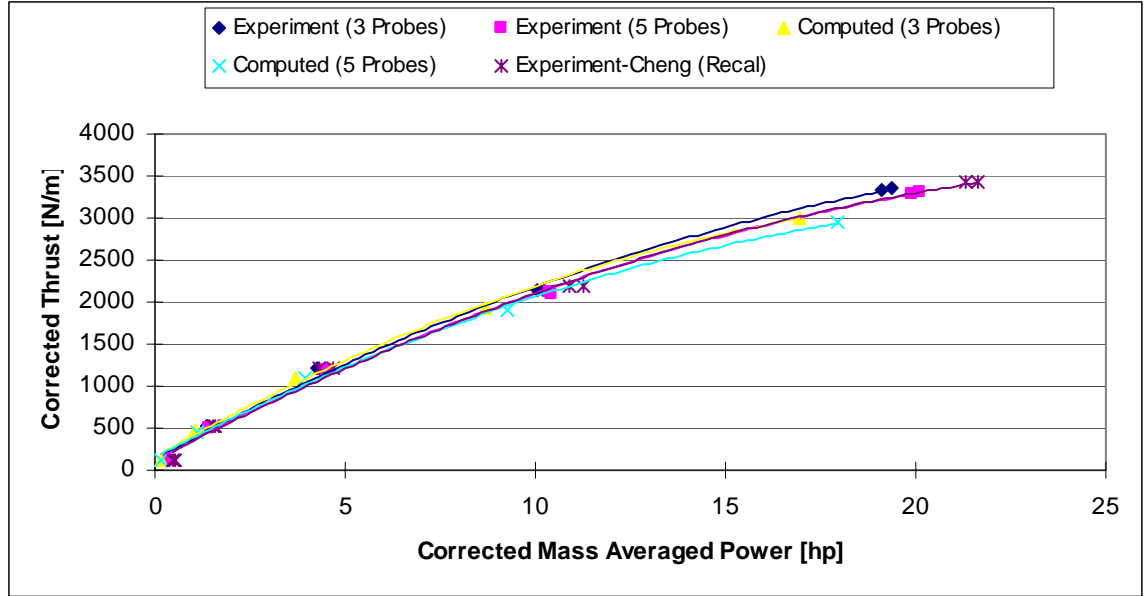


Figure 31. Comparison of thrust versus power for Test Plan I

#### F. OFF-DESIGN PERFORMANCE AT VARIOUS ROTATIONAL SPEEDS

First, the results of Test Plan II are presented and compared with the recalculated data of Cheng [Ref. 4]. In Figure 32-37, the CFF constant speed characteristics are shown based on the use of 5 probe measurements at the outlet plane. It can be seen that, which the levels of pressure ratio and temperature ratio were fairly similar at the different speeds, the range of mass flow rates from open to closed throttle were quite different, particularly at the lower speeds. At full open and stall throttle setting, the mass flow rate measured at 2000RPM was 15.25% and 77.54% less, respectively. At the higher rotational speed of 5000RPM, the mass flow rate measurements compared more favorably, however, the present measurements were down by 7.69% and 43.93% at open throttle and stall respectively.

A likely explanation for the discrepancy in mass flow rate measurements was that more accurate pressure transducers were used in the present study; i.e. the digital sensor array. To confirm this, an uncertainty analysis was performed. For Test Plan I, the uncertainties at 1000 RPM and 5000RPM were 3.72% and 0.16%, respectively, or averaged uncertainty, 1.94%. For Test Plan II, the uncertainty was 18.62% at the stall throttle position at 2000RPM, and the uncertainty was 0.13% at the full open throttle

position at 5000RPM. The overall average uncertainty was 2.39%. Detailed equations for uncertainty are given in APPENDIX C.

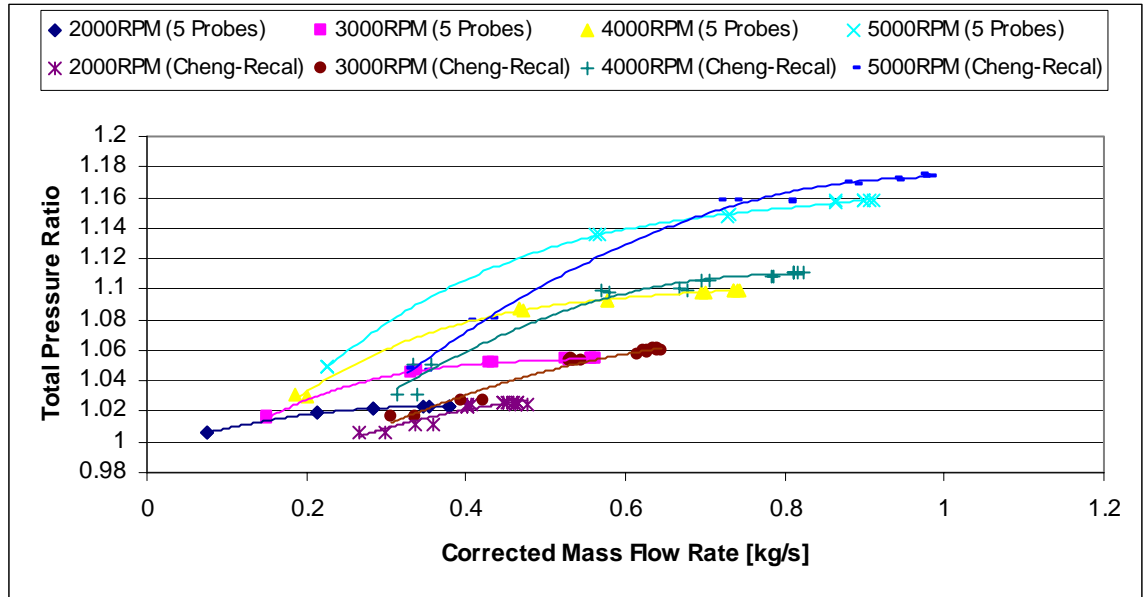


Figure 32. Comparison of total pressure ratio versus mass flow rate for Test Plan II

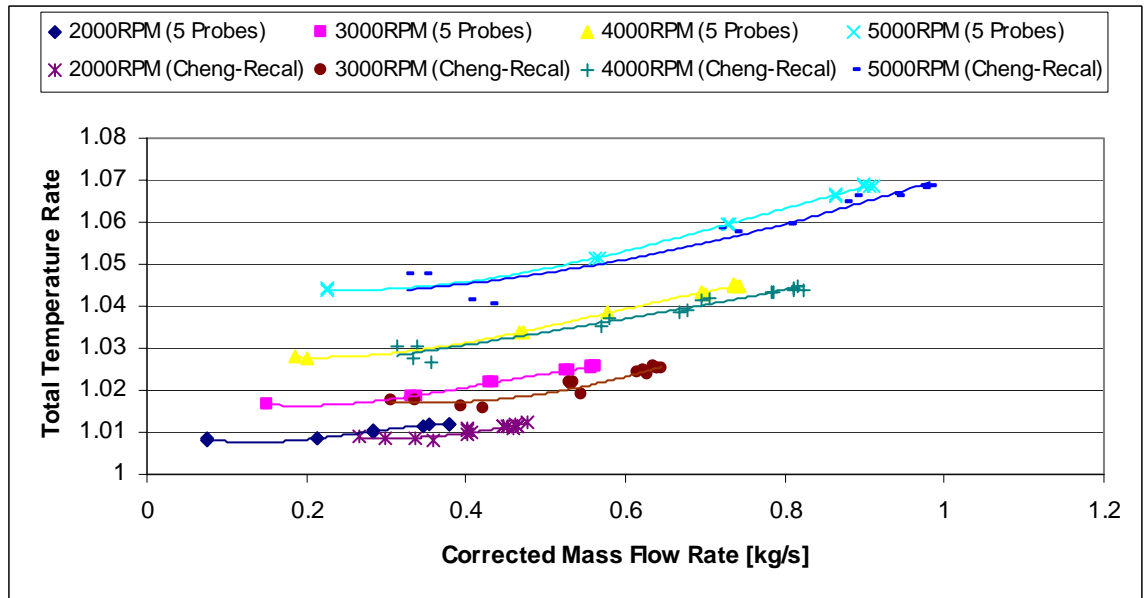


Figure 33. Comparison of total temperature ratio versus mass flow rate for Test Plan II

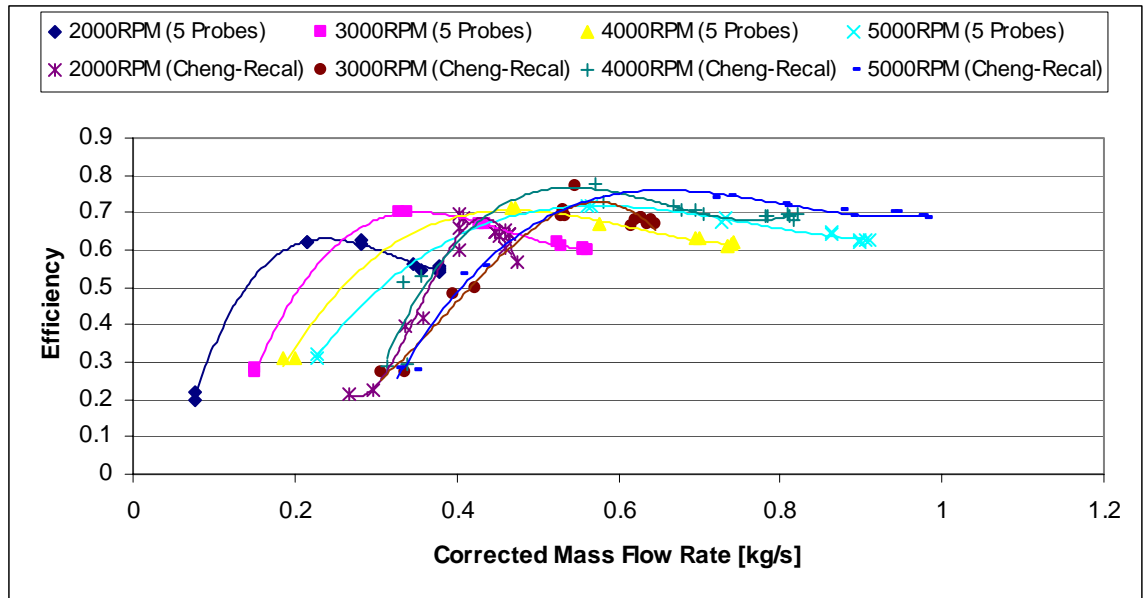


Figure 34. Comparison of efficiency versus mass flow rate for Test Plan II

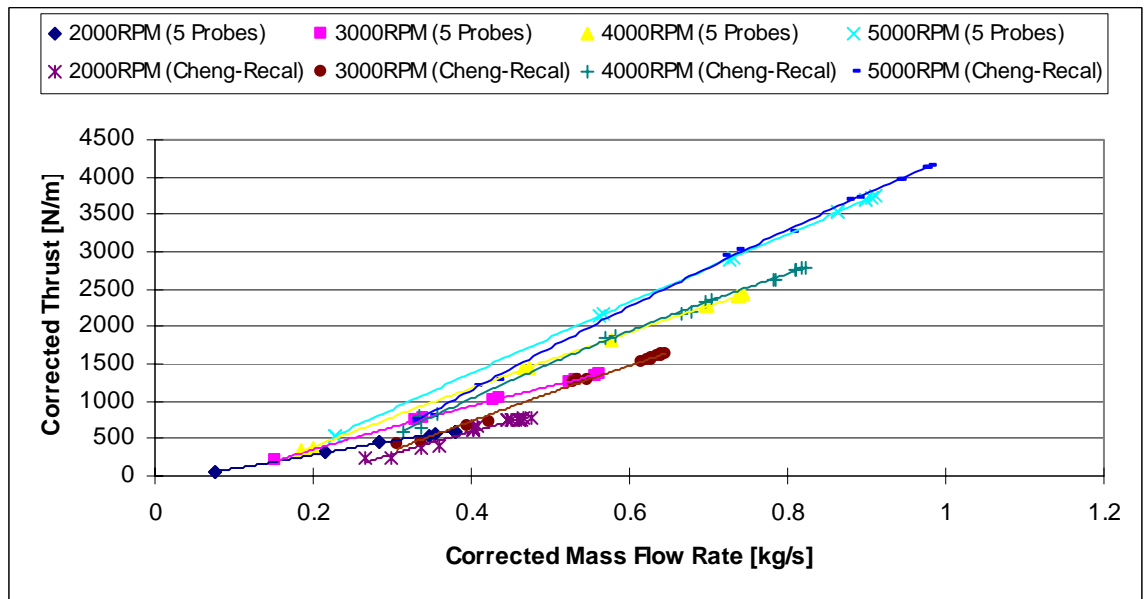


Figure 35. Comparison of thrust versus mass flow rate for Test Plan II

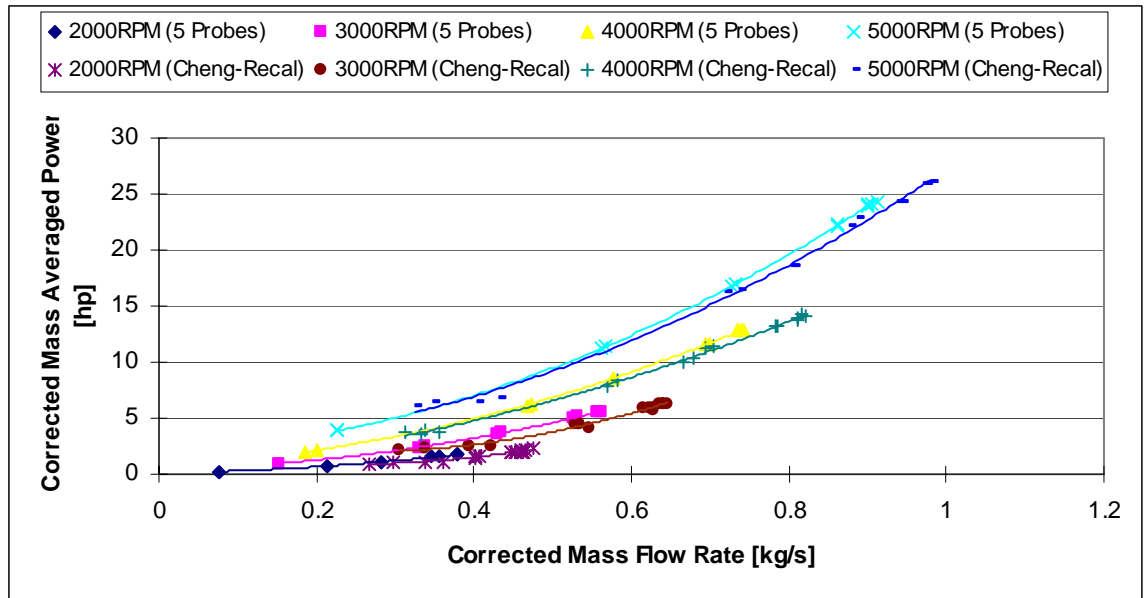


Figure 36. Comparison of power versus mass flow rate for Test Plan II

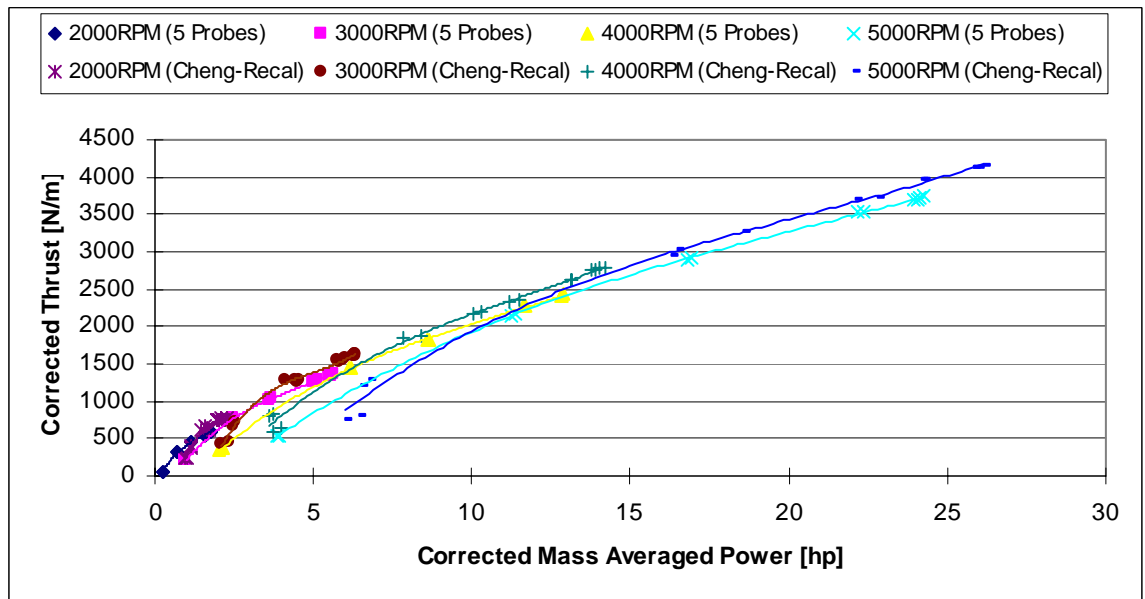


Figure 37. Comparison of thrust versus power for Test Plan II

Next, the combined performance of the baseline configuration (Test Plan I) and throttling test cases (Test Plan II) are shown together in Figures 38–40. As to be expected, the baseline characteristic intersected the constant speed characteristic curves. The exit diffuser extended the performance of the CFF by increasing its open-throttle mass flow rate and the throttle varied the performance up to a stall condition at the low mass flow rates. Peak efficiency occurred after closing the exhaust duct a certain amount at each speed. The peak efficiency was within 62% to 72%, and it increased with increasing speed over the range of speeds tested.

As discussed earlier, the mass flow rate measured in Cheng [Ref. 4] for Test Plan II was higher than that in the present experimental results. If the same combined performance were plotted, there would be no intersection between the two configurations, which supports the questioning of the previous data.

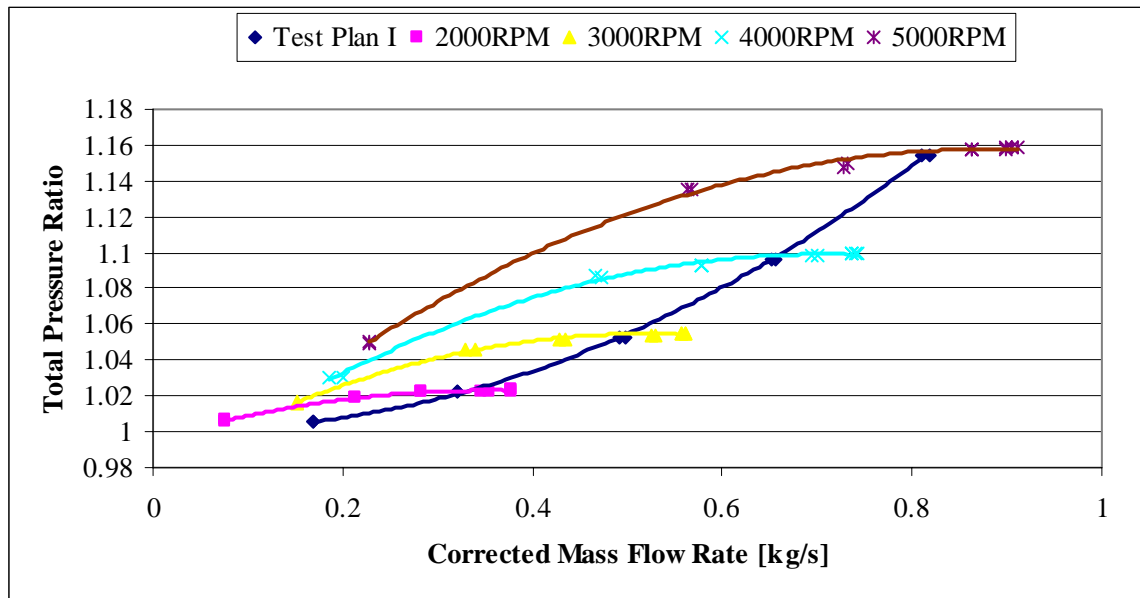


Figure 38. Combination plot of Test Plan I and Test Plan II for total pressure ratio versus mass flow rate

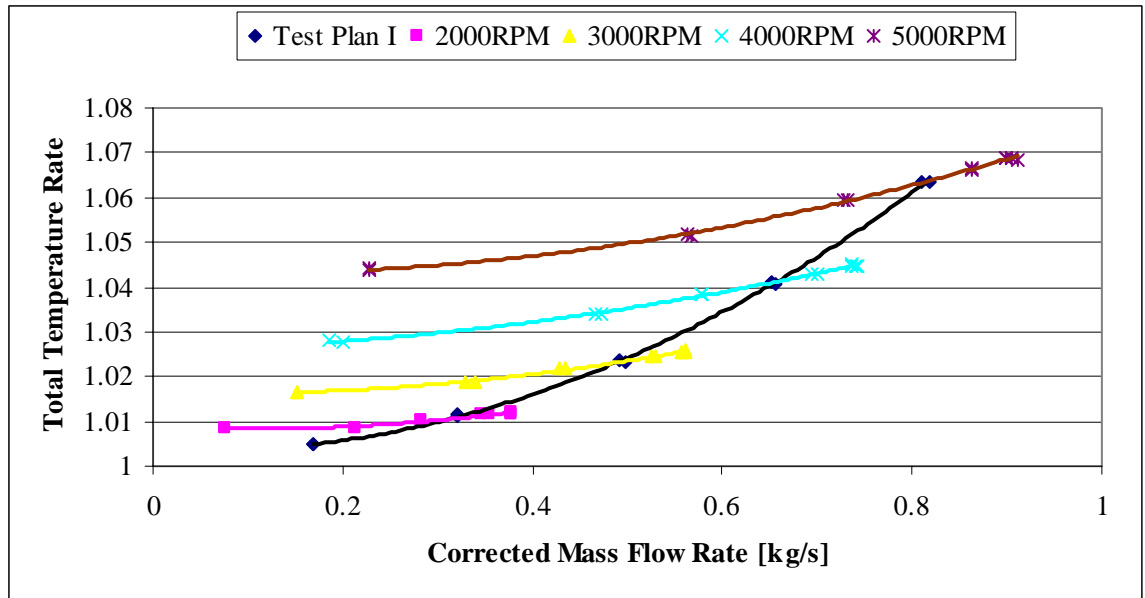


Figure 39. Combination plot of Test Plan I and Test Plan II for total temperature ratio versus mass flow rate

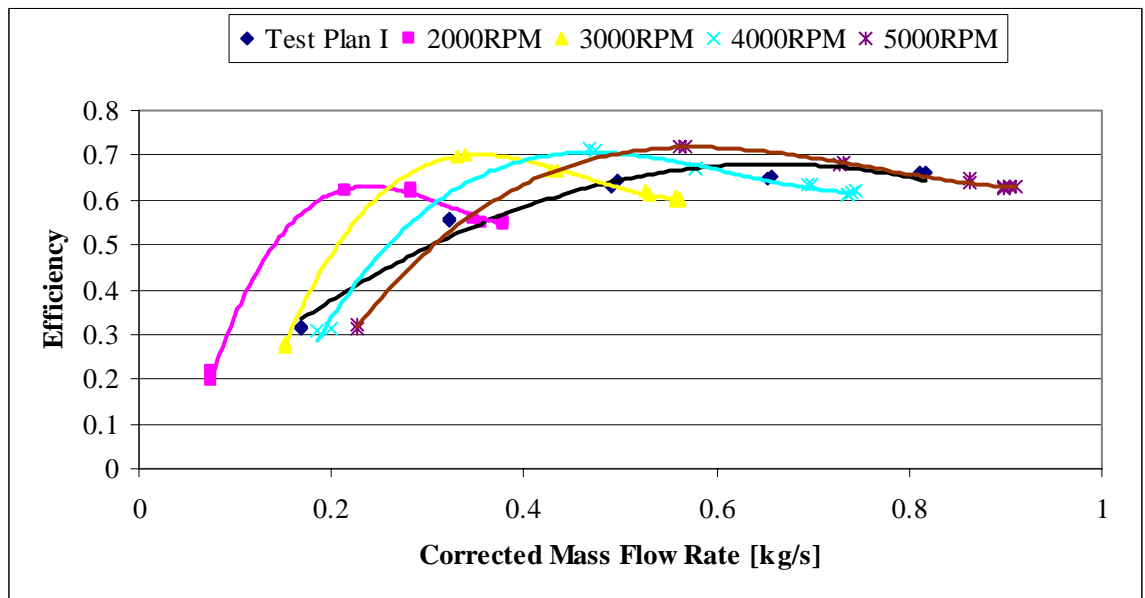


Figure 40. Combination plot of Test Plan I and Test Plan II for efficiency versus mass flow rate

Finally, the present experimental results (Test Plan II) were compared to the computed solutions (Simulation Plan II) as shown in Figures 41-43. The predicted characteristic curves were calculated in the same manner as the experiments except that the mass flow rate in the simulation was obtained directly from the inlet boundary. The numerical solutions for each specific rotational speed followed its corresponding experimental characteristic curve.

Figure 41 shows that the comparison of total pressure ratio versus mass flow rate. For the rotational speed of 2000RPM and 3000RPM, the performance was well predicted. At 5000RPM, it was observed that the difference at the open throttle position was the largest; however, the flow rate prediction was only 2.06% less than the experiment. Near stall the predictions were also very good, particularly since the calculations tended to become unstable. The data presented at the lowest mass flow rates at each speed was the last stable numerical point. Figure 42 shows the comparison of total temperature ratio versus mass flow rate. Although the predicted curves qualitatively followed the experimental curves, the total temperature ratio predicted was lower when stall occurred. The lack of agreement at low flow rates would indicate that the boundary layers within the flow-field were not adequately resolved. Figure 43 shows the efficiency predicted for different rotational speeds. At 5000 RPM the predicted peak efficiency of 72% was very close to that measured and the disagreement in efficiency was largest at open throttle and near stall. The predicted peak efficiency at 2000RPM was off by 15%; however the overall trend was reasonable. At open throttle, the differences between calculation and experiment were 6% and 20% for 2000RPM and 5000RPM respectively. The calculations also showed that the CFF could be run at larger diffuser angles; i.e. higher mass flow rates.



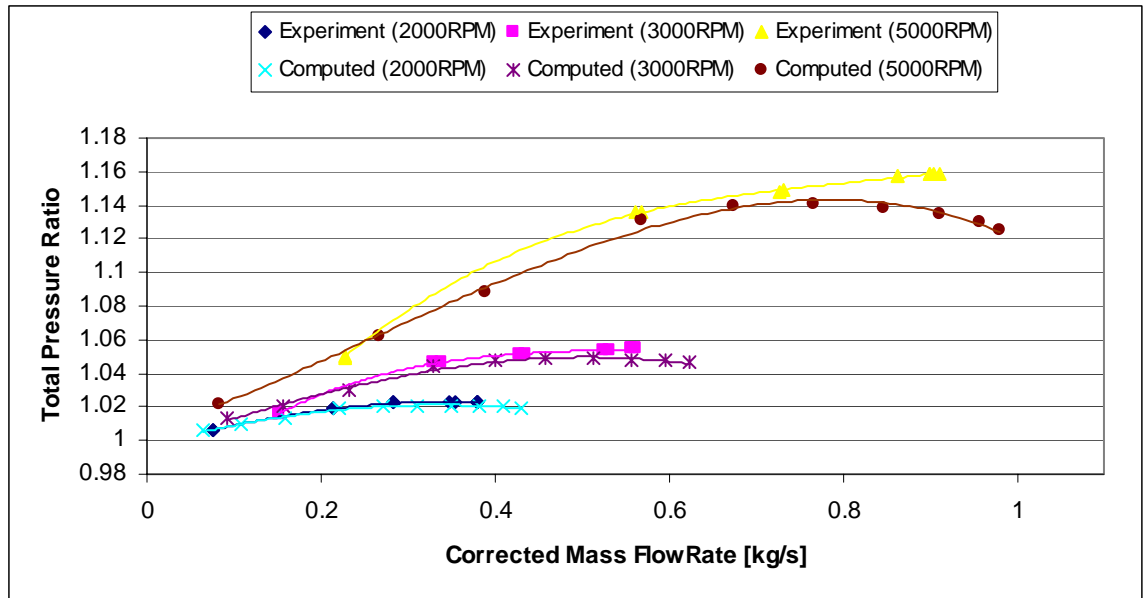


Figure 41. Comparison of experimental and computed results for total pressure ratio versus mass flow rate

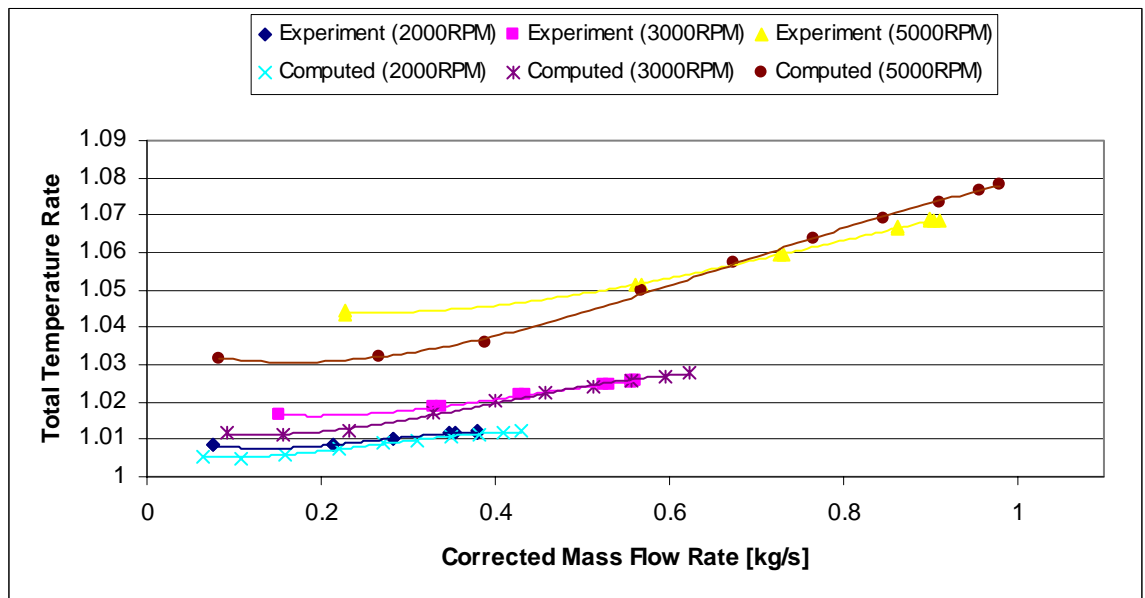


Figure 42. Comparison of experimental and computed results for total temperature ratio versus mass flow rate

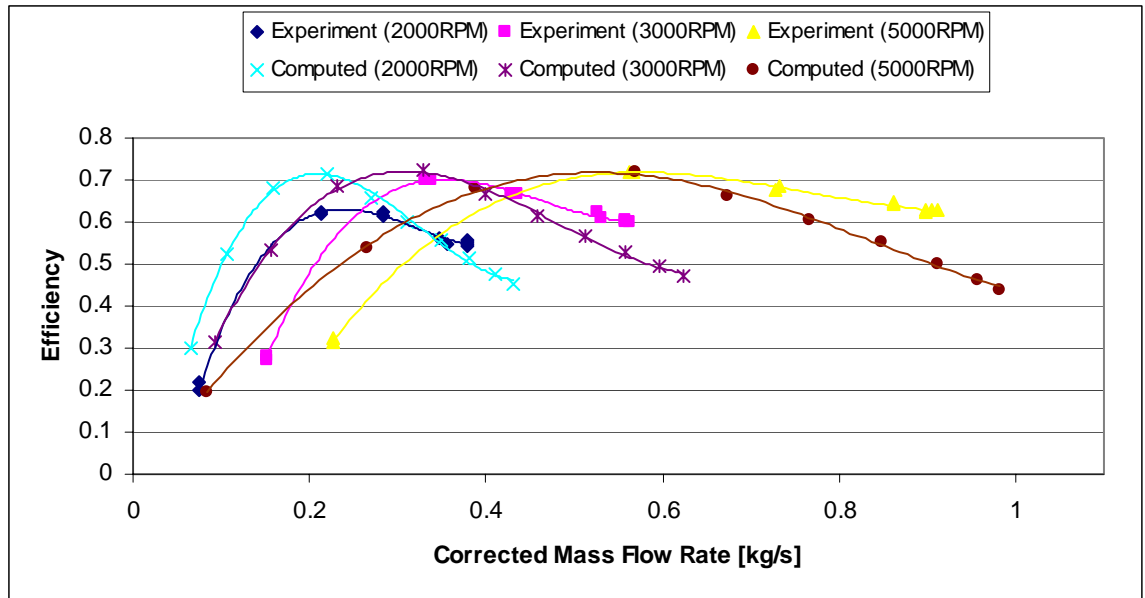


Figure 43. Comparison of experimental and computed results for efficiencies versus mass flow rate

## V. SUMMARY AND CONCLUSIONS

The CFD study was initially carried out with the intension of predicting the performance data of Cheng [Ref. 4]. Results of the prediction of the throttling configurations raised the question of the accuracy in the previous experiments. An improved experiment was therefore undertaken as reporeted here. When compared with accurate experimental data, the overall off-design performance of the CFF was numerically well predicted. Specifically, the comparison of computed and experimental results was made by employing the same calculating procedure for both. The study also indicated that the standard day conditions would be a better boundary condition when setting up the model. Moreover, the consistency study supported the contention of some error in the deterimentation of mass flow rate in Cheng [Ref. 4]. The Mach number distribution in the flow field at 3000RPM at stall is shown in Figure 44. The flow entrained into the CFF moved closer to the HPC, while the vortex around the LPC became larger, pushing the flow to the left. The zone of lower velocity increased and the mass flow rate was hence reduced. A dramatic drop-off in efficiency occurred because of the attendant viscous mixing losses.

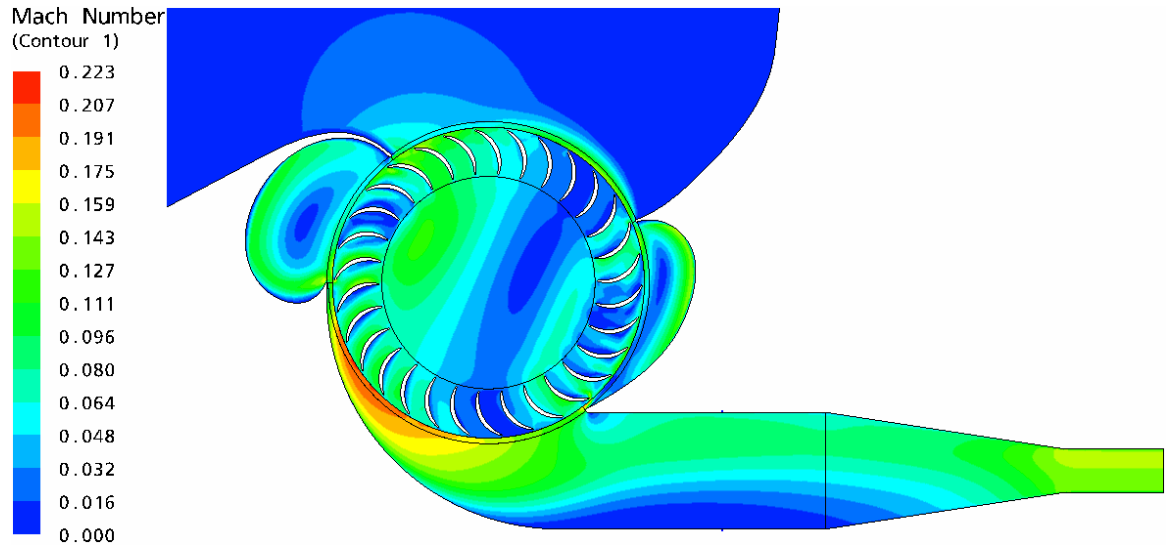
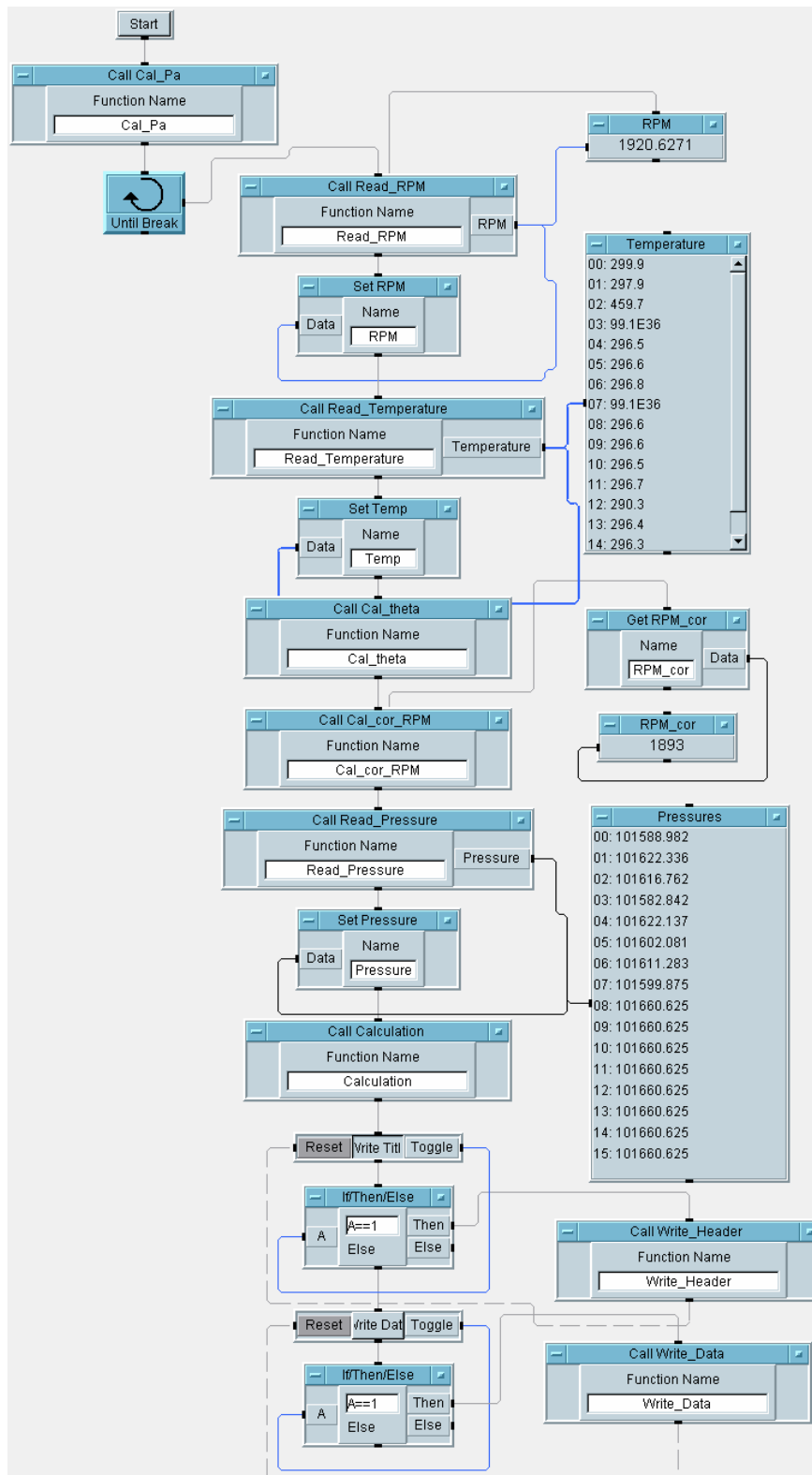


Figure 44. Distribution of Mach at 3000RPM with stall position setting

With the present successful modeling, more configurations of the 12-inch rotor can be computed and compared to the experimental results, particularly at higher

rotational speeds and higher exit diffuser angles. Furthermore, a case study of the 6-inch rotor with various spanwise lengths should be computed using the procedures presented.

## APPENDIX A. CFF\_DAQ STRUCTURAL TREE

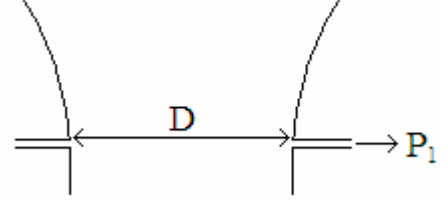


THIS PAGE INTENTIONALLY LEFT BLANK

## APPENDIX B. MASS FLOW RATE MEASUREMENT OF THE CFF

Mass flow rate can be obtained using the continuity equation and introducing the equation of state; it is then

$$\dot{m} = \frac{P_1}{RT_1} AV_{in}$$



where  $P_1$  is measured directly from static tapes at bell-mouth;  $A$  is the cross-sectional area of the bell-mouth,  $A = \pi D^2 / 4$ ,  $D = 6.25''$ ;  $R$  is gas constant. However,  $T_1$  and  $V_{in}$  are unknown and have to be calculated from parameters which can be measured. The procedure for finding  $T_1$  and  $V_{in}$  is as follows:

From steady flow energy equation for a perfect gas;

$$T_{t1} = T_1 + \frac{V_{in}^2}{2c_p}, \text{ where } c_p = \frac{\gamma R}{\gamma - 1}$$

then,  $T_1$  is obtained as

$$T_1 = T_{t1} - \frac{(\gamma - 1)}{2\gamma R} V_{in}^2$$

From Bernoulli's equation (which assumes small density change), and the equation of state,

$$P_{t1} = P_{atm} = P_1 + \frac{1}{2} \rho V_{in}^2$$

$$P_{t1} = P_1 + \frac{1}{2} \frac{P_1}{RT_1} V^2 = P_1 \left( 1 + \frac{V_{in}^2}{2RT_1} \right)$$

By introducing  $T_1$ ,

$$\frac{P_{t1}}{P_1} = PR = \left\{ 1 + \frac{V_{in}^2}{2R \left[ T_{t1} - \left( \frac{\gamma - 1}{2\gamma R} \right) V_{in}^2 \right]} \right\}$$

Rearrange equation

$$PR = \frac{2R \left[ T_{t1} - \left( \frac{\gamma-1}{2\gamma R} \right) V_{in}^2 \right] + V_{in}^2}{2R \left[ T_{t1} - \left( \frac{\gamma-1}{2\gamma R} \right) V_{in}^2 \right]}$$

$$PR \left\{ 2R \left[ T_{t1} - \left( \frac{\gamma-1}{2\gamma R} \right) V_{in}^2 \right] \right\} = 2R \left[ T_{t1} - \left( \frac{\gamma-1}{2\gamma R} \right) V_{in}^2 \right] + V_{in}^2$$

$$\left[ \frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) PR \right] V_{in}^2 = 2RT_{t1}(PR-1)$$

Therefore,  $V_{in}$  is obtained as

$$V_{in} = \sqrt{\frac{2RT_{t1}(PR-1)}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) PR}}$$



## APPENDIX C. UNCERTAINTY STUDY

In APPENDIX B, the description of derivation of mass flow was made, and it can be expressed in the form of

$$\dot{m} = \frac{P_1}{RT_1(T_{t1}, P_{t1}, P_1)} AV_{in}(T_{t1}, P_{t1}, P_1) \quad (C.01)$$

$T_{t1}$ ,  $P_{t1}$ ,  $P$  and  $A$  are independent variables and measured from instruments, whose corresponding uncertainties are provided in Table C.01

Variable	Accuracy, $w$
$T_{t1}$	$\pm 1^\circ C$ ( $w_{T_{t1}}$ )
$P_{t1}$	$\pm 0.001 bar$ ( $w_{P_{t1}}$ )
$P_1$	$\pm 0.05\%$ ( $w_{P_1}$ )
$A$	$\pm 0.0001 \pi in^2$ ( $w_A$ )

Table C.01 Uncertainty in each measurement

Let  $w_r$  be the uncertainty in the mass flow rate, and then it is given as

$$w_r = \left[ \left( \frac{\partial \dot{m}}{\partial T_{t1}} w_{T_{t1}} \right)^2 + \left( \frac{\partial \dot{m}}{\partial P_{t1}} w_{P_{t1}} \right)^2 + \left( \frac{\partial \dot{m}}{\partial P_1} w_{P_1} \right)^2 + \left( \frac{\partial \dot{m}}{\partial A} w_A \right)^2 \right]^{1/2} \quad (C.02)$$

where each  $w$  presents the uncertainty in each measurement, and their corresponding derivatives were found as

$$\frac{\partial \dot{m}}{\partial T_{t1}} = \frac{P_1 A}{RT_1} \frac{\partial V_{in}}{\partial T_{t1}} - \frac{P_1 A V_{in}}{R} \frac{1}{T_1^2} \frac{\partial T_1}{\partial T_{t1}} \quad (C.03)$$

$$\frac{\partial \dot{m}}{\partial P_{t1}} = \frac{P_1 A}{RT_1} \frac{\partial V_{in}}{\partial P_{t1}} - \frac{P_1 A V_{in}}{R} \frac{1}{T_1^2} \frac{\partial T_1}{\partial P_{t1}} \quad (C.04)$$

$$\frac{\partial \dot{m}}{\partial P_1} = \frac{A V_{in}}{RT_1} + \frac{P_1 A}{RT_1} \frac{\partial V_{in}}{\partial P_1} - \frac{P_1 A V_{in}}{R} \frac{1}{T_1^2} \frac{\partial T_1}{\partial P_1} \quad (C.05)$$

$$\frac{\partial \dot{m}}{\partial A} = \frac{P_1 V_{in}}{RT_1} \quad (C.06)$$

Now, there are six more derivatives to be determined,

$$\frac{\partial V_{in}}{\partial T_{t1}}, \frac{\partial V_{in}}{\partial P_{t1}}, \frac{\partial V_{in}}{\partial P_1}, \frac{\partial T_1}{\partial T_{t1}}, \frac{\partial T_1}{\partial P_{t1}}, \frac{\partial T_1}{\partial P_1}$$

These six derivatives were derived as follows:

$$\frac{\partial V_{in}}{\partial T_{t1}} = \frac{\partial}{\partial T_{t1}} \left[ \frac{2RT_{t1} \left( \frac{P_{t1}}{P_1} - 1 \right)}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1}} \right]^{1/2} = \frac{1}{2} \left[ \frac{2RT_{t1} \left( \frac{P_{t1}}{P_1} - 1 \right)}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1}} \right]^{-1/2} \left[ \frac{2R \left( \frac{P_{t1}}{P_1} - 1 \right)}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1}} \right] \quad (C.07)$$

$$\begin{aligned} \frac{\partial V_{in}}{\partial P_{t1}} &= \frac{\partial}{\partial P_{t1}} \left[ \frac{2RT_{t1} \left( \frac{P_{t1}}{P_1} - 1 \right)}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1}} \right]^{1/2} \\ &= \frac{1}{2} \left[ \frac{2RT_{t1} \left( \frac{P_{t1}}{P_1} - 1 \right)}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1}} \right]^{-1/2} \left\{ \frac{2RT_{t1} \frac{1}{P_1}}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1}} - \frac{2RT_{t1} \left( \frac{P_{t1}}{P_1} - 1 \right) \left( \frac{\gamma-1}{2} \right) \frac{1}{P_1}}{\left[ \frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1} \right]^2} \right\} \end{aligned} \quad (C.08)$$

$$\begin{aligned} \frac{\partial V_{in}}{\partial P_1} &= \frac{\partial}{\partial P_1} \left[ \frac{2RT_{t1} \left( \frac{P_{t1}}{P_1} - 1 \right)}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1}} \right]^{1/2} \\ &= \frac{1}{2} \left[ \frac{2RT_{t1} \left( \frac{P_{t1}}{P_1} - 1 \right)}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1}} \right]^{-1/2} \left\{ \frac{2RT_{t1} \left( \frac{P_{t1}}{P_1} - 1 \right) \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1^2}}{\left[ \frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1} \right]^2} - \frac{2RT_{t1} \frac{P_{t1}}{P_1^2}}{\frac{1}{\gamma} + \left( \frac{\gamma-1}{2} \right) \frac{P_{t1}}{P_1}} \right\} \end{aligned} \quad (C.09)$$

$$\frac{\partial T_1}{\partial T_{t1}} = \frac{\partial}{\partial T_{t1}} \left( T_{t1} - \frac{\gamma-1}{2\gamma R} V_{in}^2 \right) = 1 - \left( \frac{\gamma-1}{2\gamma R} \right) (2V_{in}) \left( \frac{\partial V_{in}}{\partial T_{t1}} \right) \quad (C.10)$$

$$\frac{\partial T_1}{\partial P_{t1}} = \frac{\partial}{\partial P_{t1}} \left( T_{t1} - \frac{\gamma-1}{2\gamma R} V_{in}^2 \right) = - \left( \frac{\gamma-1}{2\gamma R} \right) (2V_{in}) \left( \frac{\partial V_{in}}{\partial P_{t1}} \right) \quad (C.11)$$

$$\frac{\partial T_1}{\partial P_1} = \frac{\partial}{\partial P_1} \left( T_{t1} - \frac{\gamma-1}{2\gamma R} V_{in}^2 \right) = - \left( \frac{\gamma-1}{2\gamma R} \right) (2V_{in}) \left( \frac{\partial V_{in}}{\partial P_1} \right) \quad (C.12)$$

Substituting Eq. (C.07), (C.08) and (C.09) into Eq. (C.10), (C.11) and (C.12), respectively, then all derivatives were expressed in terms of independent variables.

Once all derivatives required were found, substitute them back to Eq. (C.02), then the uncertainty in the mass flow rate was determined.

THIS PAGE INTENTIONALLY LEFT BLANK

## APPENDIX D. GEOMETRY GENERATION IN ICEM-CFD

### 1. Starting a New Project

#### a) Launch ANSYS ICEMCFD-CFX


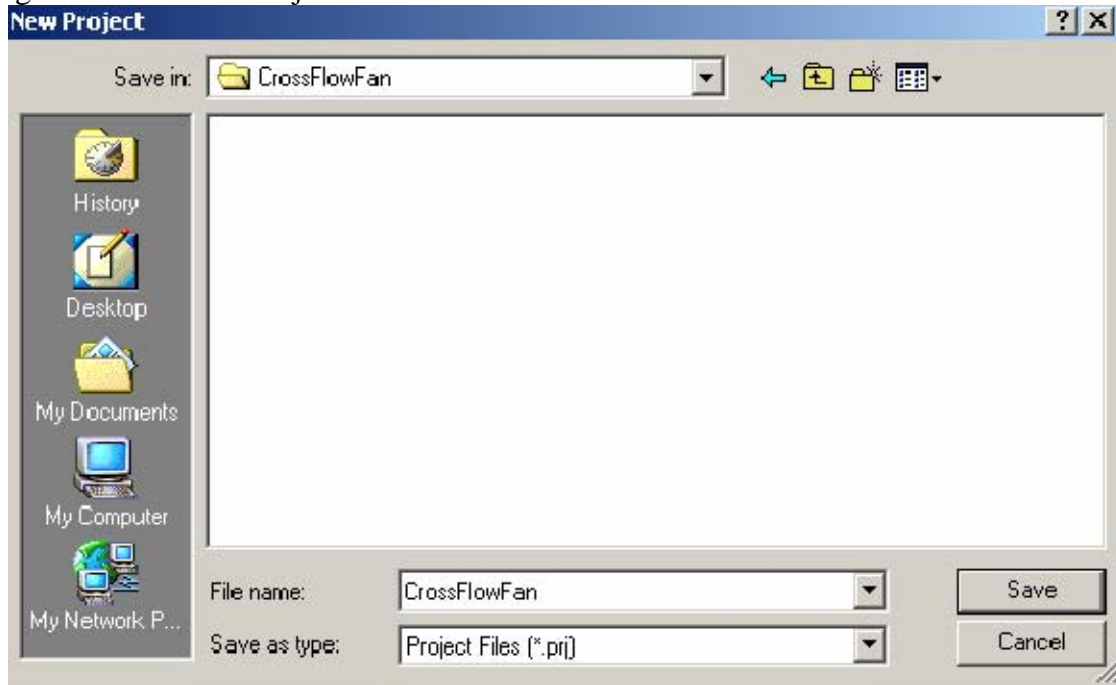
Select **File > New Project** from the Main menu and click on  (Create New Directory) and enter Directory name as in Figure D01.

Figure D01: New Project Window



### 2. 2-D Model Generation

#### 2.1. Blade generation

##### a) Point Creation

**Geometry > Create Point > Explicit Coordinates:** Select the  (Explicit coordinate) to open the *Explicit Location* window as shown in Figure D02.

As shown in Figure D02, Select **Create 1 point**, and assign coordinates (0 0 0). Input the **Part** name POINTS, and the **Name** as POINTS.00 and press **Apply** to create a point. This is a reference point *when Import Geometry* is required.

Figure D02: Point creation window

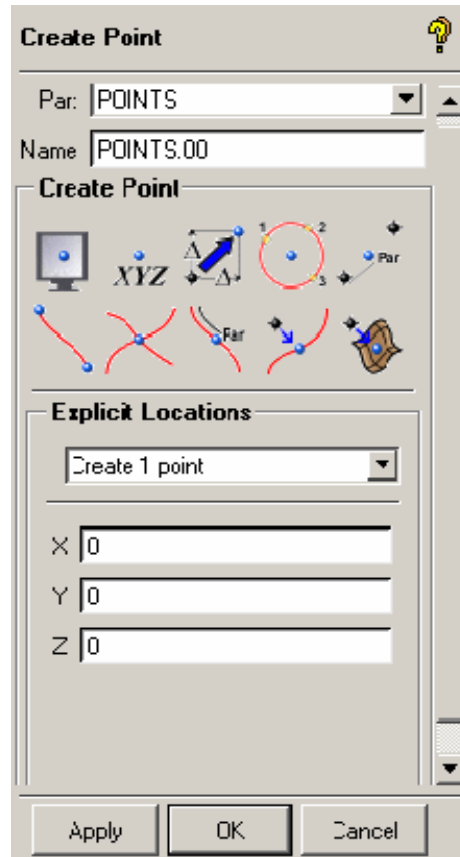
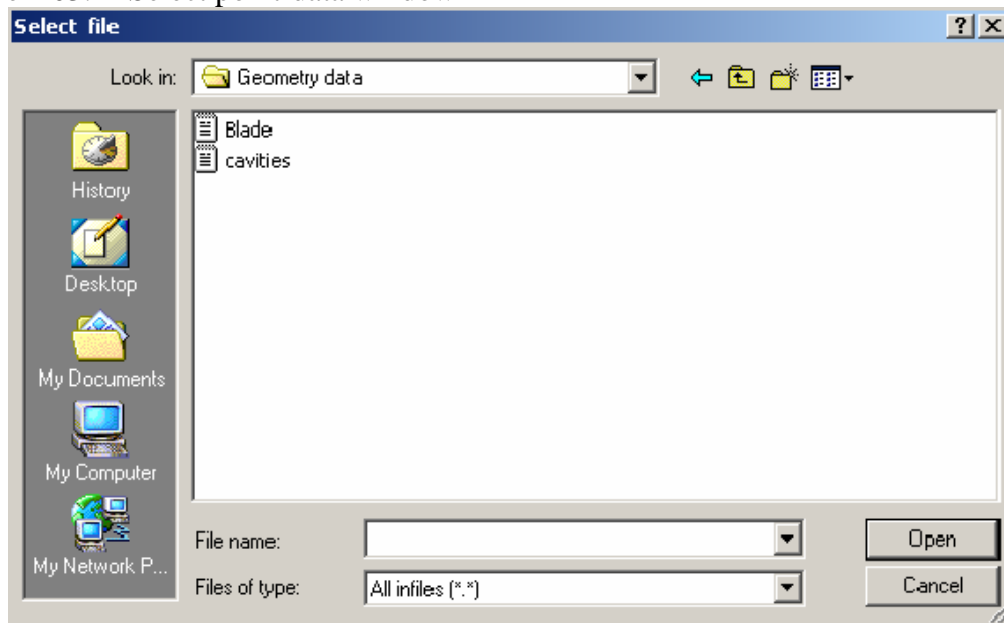


Figure D03: Select point-data window



Select **File > Import Geometry > Formatted point data** from the Main menu and choose **Blade.txt** as shown in Figure D03. Press **Open**, the window shown as Figure D04

appears. Uncheck **Import Curves**, **Import Surfaces** and **Plot3D Format**, Then Set **Approximation Tolerance** to 0. Keep the name of **Point Part** as default. To accept the points by click **Apply**, then it is read and shown in Figure D05.

Figure D04: Import formatted point data window

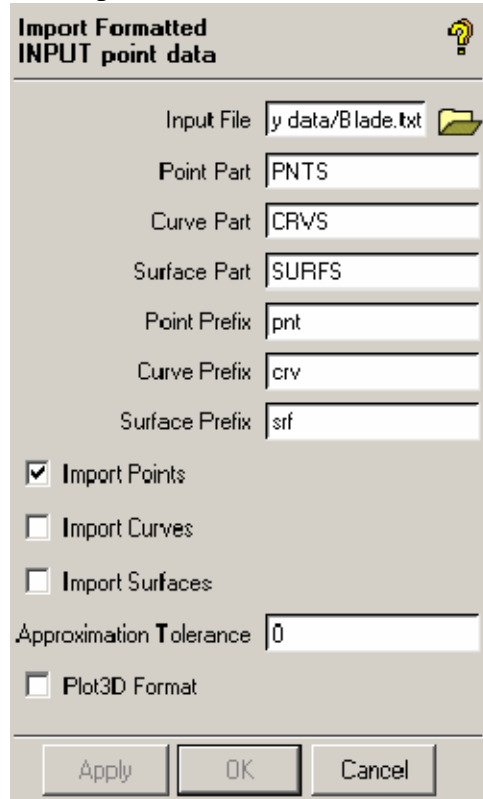


Figure D05: Points after importing

pnt1  
pnt2  
pnt3  
pnt4  
pnt5  
pnt6  
pnt7  
pnt8  
pnt9  
pnt10  
pnt11  
pnt12  
pnt13  
pnt14  
pnt15  
pnt16  
pnt17  
pnt18  
pnt19  
pnt20  
pnt21  
pnt22  
pnt23  
pnt24  
pnt25  
pnt26  
pnt27  
pnt28  
pnt29  
pnt30  
pnt31  
pnt32  
pnt33  
pnt34  
pnt35  
pnt36  
pnt37  
pnt38  
pnt39  
pnt40  
pnt41  
pnt42  
pnt43  
pnt44  
pnt45  
pnt46  
pnt47  
pnt48  
pnt49  
pnt50  
pnt51  
pnt52  
pnt53  
pnt54  
pnt55  
pnt56  
pnt57  
pnt58  
pnt59  
pnt60  
pnt61  
pnt62  
pnt63  
pnt64  
pnt65  
pnt66  
pnt67  
pnt68  
pnt69  
pnt70  
pnt71  
pnt72  
pnt73  
pnt74  
pnt75  
pnt76  
pnt77  
pnt78  
pnt79  
pnt80  
pnt81  
pnt82  
pnt83  
pnt84  
pnt85  
pnt86  
pnt87  
pnt88  
pnt89  
pnt90  
pnt91  
pnt92  
pnt93  
pnt94  
pnt95  
pnt96  
pnt97  
pnt98  
pnt99  
pnt100  
pnt101  
pnt102  
pnt103  
pnt104  
pnt105  
pnt106  
pnt107  
pnt108  
pnt109  
pnt110  
pnt111  
pnt112  
pnt113  
pnt114  
pnt115  
pnt116  
pnt117  
pnt118  
pnt119  
pnt120  
pnt121  
pnt122  
pnt123  
pnt124  
pnt125  
pnt126  
pnt127  
pnt128  
pnt129  
pnt130  
pnt131  
pnt132  
pnt133  
pnt134  
pnt135  
pnt136  
pnt137  
pnt138  
pnt139  
pnt140  
pnt141  
pnt142  
pnt143  
pnt144  
pnt145  
pnt146  
pnt147  
pnt148  
pnt149  
pnt150  
pnt151  
pnt152  
pnt153  
pnt154  
pnt155  
pnt156  
pnt157  
pnt158  
pnt159  
pnt160  
pnt161  
pnt162  
pnt163  
pnt164  
pnt165  
pnt166  
pnt167  
pnt168  
pnt169  
pnt170  
pnt171  
pnt172  
pnt173  
pnt174  
pnt175  
pnt176  
pnt177  
pnt178  
pnt179  
pnt180  
pnt181  
pnt182  
pnt183  
pnt184  
pnt185  
pnt186  
pnt187  
pnt188  
pnt189  
pnt190  
pnt191  
pnt192  
pnt193  
pnt194  
pnt195  
pnt196  
pnt197  
pnt198  
pnt199  
pnt200  
pnt201  
pnt202  
pnt203  
pnt204  
pnt205  
pnt206  
pnt207  
pnt208  
pnt209  
pnt210  
pnt211  
pnt212  
pnt213  
pnt214  
pnt215  
pnt216  
pnt217  
pnt218  
pnt219  
pnt220  
pnt221  
pnt222  
pnt223  
pnt224  
pnt225  
pnt226  
pnt227  
pnt228  
pnt229  
pnt230  
pnt231  
pnt232  
pnt233  
pnt234  
pnt235  
pnt236  
pnt237  
pnt238  
pnt239  
pnt240  
pnt241  
pnt242  
pnt243  
pnt244  
pnt245  
pnt246  
pnt247  
pnt248  
pnt249  
pnt250  
pnt251  
pnt252  
pnt253  
pnt254  
pnt255  
pnt256  
pnt257  
pnt258  
pnt259  
pnt260  
pnt261  
pnt262  
pnt263  
pnt264  
pnt265  
pnt266  
pnt267  
pnt268  
pnt269  
pnt270  
pnt271  
pnt272  
pnt273  
pnt274  
pnt275  
pnt276  
pnt277  
pnt278  
pnt279  
pnt280  
pnt281  
pnt282  
pnt283  
pnt284  
pnt285  
pnt286  
pnt287  
pnt288  
pnt289  
pnt290  
pnt291  
pnt292  
pnt293  
pnt294  
pnt295  
pnt296  
pnt297  
pnt298  
pnt299  
pnt300  
pnt301  
pnt302  
pnt303  
pnt304  
pnt305  
pnt306  
pnt307  
pnt308  
pnt309  
pnt310  
pnt311  
pnt312  
pnt313  
pnt314  
pnt315  
pnt316  
pnt317  
pnt318  
pnt319  
pnt320  
pnt321  
pnt322  
pnt323  
pnt324  
pnt325  
pnt326  
pnt327  
pnt328  
pnt329  
pnt330  
pnt331  
pnt332  
pnt333  
pnt334  
pnt335  
pnt336  
pnt337  
pnt338  
pnt339  
pnt340  
pnt341  
pnt342  
pnt343  
pnt344  
pnt345  
pnt346  
pnt347  
pnt348  
pnt349  
pnt350  
pnt351  
pnt352  
pnt353  
pnt354  
pnt355  
pnt356  
pnt357  
pnt358  
pnt359  
pnt360  
pnt361  
pnt362  
pnt363  
pnt364  
pnt365  
pnt366  
pnt367  
pnt368  
pnt369  
pnt370  
pnt371  
pnt372  
pnt373  
pnt374  
pnt375  
pnt376  
pnt377  
pnt378  
pnt379  
pnt380  
pnt381  
pnt382  
pnt383  
pnt384  
pnt385  
pnt386  
pnt387  
pnt388  
pnt389  
pnt390  
pnt391  
pnt392  
pnt393  
pnt394  
pnt395  
pnt396  
pnt397  
pnt398  
pnt399  
pnt400  
pnt401  
pnt402  
pnt403  
pnt404  
pnt405  
pnt406  
pnt407  
pnt408  
pnt409  
pnt410  
pnt411  
pnt412  
pnt413  
pnt414  
pnt415  
pnt416  
pnt417  
pnt418  
pnt419  
pnt420  
pnt421  
pnt422  
pnt423  
pnt424  
pnt425  
pnt426  
pnt427  
pnt428  
pnt429  
pnt430  
pnt431  
pnt432  
pnt433  
pnt434  
pnt435  
pnt436  
pnt437  
pnt438  
pnt439  
pnt440  
pnt441  
pnt442  
pnt443  
pnt444  
pnt445  
pnt446  
pnt447  
pnt448  
pnt449  
pnt450  
pnt451  
pnt452  
pnt453  
pnt454  
pnt455  
pnt456  
pnt457  
pnt458  
pnt459  
pnt460  
pnt461  
pnt462  
pnt463  
pnt464  
pnt465  
pnt466  
pnt467  
pnt468  
pnt469  
pnt470  
pnt471  
pnt472  
pnt473  
pnt474  
pnt475  
pnt476  
pnt477  
pnt478  
pnt479  
pnt480  
pnt481  
pnt482  
pnt483  
pnt484  
pnt485  
pnt486  
pnt487  
pnt488  
pnt489  
pnt490  
pnt491  
pnt492  
pnt493  
pnt494  
pnt495  
pnt496  
pnt497  
pnt498  
pnt499  
pnt500  
pnt501  
pnt502  
pnt503  
pnt504  
pnt505  
pnt506  
pnt507  
pnt508  
pnt509  
pnt510  
pnt511  
pnt512  
pnt513  
pnt514  
pnt515  
pnt516  
pnt517  
pnt518  
pnt519  
pnt520  
pnt521  
pnt522  
pnt523  
pnt524  
pnt525  
pnt526  
pnt527  
pnt528  
pnt529  
pnt530  
pnt531  
pnt532  
pnt533  
pnt534  
pnt535  
pnt536  
pnt537  
pnt538  
pnt539  
pnt540  
pnt541  
pnt542  
pnt543  
pnt544  
pnt545  
pnt546  
pnt547  
pnt548  
pnt549  
pnt550  
pnt551  
pnt552  
pnt553  
pnt554  
pnt555  
pnt556  
pnt557  
pnt558  
pnt559  
pnt560  
pnt561  
pnt562  
pnt563  
pnt564  
pnt565  
pnt566  
pnt567  
pnt568  
pnt569  
pnt570  
pnt571  
pnt572  
pnt573  
pnt574  
pnt575  
pnt576  
pnt577  
pnt578  
pnt579  
pnt580  
pnt581  
pnt582  
pnt583  
pnt584  
pnt585  
pnt586  
pnt587  
pnt588  
pnt589  
pnt590  
pnt591  
pnt592  
pnt593  
pnt594  
pnt595  
pnt596  
pnt597  
pnt598  
pnt599  
pnt600  
pnt601  
pnt602  
pnt603  
pnt604  
pnt605  
pnt606  
pnt607  
pnt608  
pnt609  
pnt610  
pnt611  
pnt612  
pnt613  
pnt614  
pnt615  
pnt616  
pnt617  
pnt618  
pnt619  
pnt620  
pnt621  
pnt622  
pnt623  
pnt624  
pnt625  
pnt626  
pnt627  
pnt628  
pnt629  
pnt630  
pnt631  
pnt632  
pnt633  
pnt634  
pnt635  
pnt636  
pnt637  
pnt638  
pnt639  
pnt640  
pnt641  
pnt642  
pnt643  
pnt644  
pnt645  
pnt646  
pnt647  
pnt648  
pnt649  
pnt650  
pnt651  
pnt652  
pnt653  
pnt654  
pnt655  
pnt656  
pnt657  
pnt658  
pnt659  
pnt660  
pnt661  
pnt662  
pnt663  
pnt664  
pnt665  
pnt666  
pnt667  
pnt668  
pnt669  
pnt670  
pnt671  
pnt672  
pnt673  
pnt674  
pnt675  
pnt676  
pnt677  
pnt678  
pnt679  
pnt680  
pnt681  
pnt682  
pnt683  
pnt684  
pnt685  
pnt686  
pnt687  
pnt688  
pnt689  
pnt690  
pnt691  
pnt692  
pnt693  
pnt694  
pnt695  
pnt696  
pnt697  
pnt698  
pnt699  
pnt700  
pnt701  
pnt702  
pnt703  
pnt704  
pnt705  
pnt706  
pnt707  
pnt708  
pnt709  
pnt710  
pnt711  
pnt712  
pnt713  
pnt714  
pnt715  
pnt716  
pnt717  
pnt718  
pnt719  
pnt720  
pnt721  
pnt722  
pnt723  
pnt724  
pnt725  
pnt726  
pnt727  
pnt728  
pnt729  
pnt730  
pnt731  
pnt732  
pnt733  
pnt734  
pnt735  
pnt736  
pnt737  
pnt738  
pnt739  
pnt740  
pnt741  
pnt742  
pnt743  
pnt744  
pnt745  
pnt746  
pnt747  
pnt748  
pnt749  
pnt750  
pnt751  
pnt752  
pnt753  
pnt754  
pnt755  
pnt756  
pnt757  
pnt758  
pnt759  
pnt760  
pnt761  
pnt762  
pnt763  
pnt764  
pnt765  
pnt766  
pnt767  
pnt768  
pnt769  
pnt770  
pnt771  
pnt772  
pnt773  
pnt774  
pnt775  
pnt776  
pnt777  
pnt778  
pnt779  
pnt780  
pnt781  
pnt782  
pnt783  
pnt784  
pnt785  
pnt786  
pnt787  
pnt788  
pnt789  
pnt790  
pnt791  
pnt792  
pnt793  
pnt794  
pnt795  
pnt796  
pnt797  
pnt798  
pnt799  
pnt800  
pnt801  
pnt802  
pnt803  
pnt804  
pnt805  
pnt806  
pnt807  
pnt808  
pnt809  
pnt810  
pnt811  
pnt812  
pnt813  
pnt814  
pnt815  
pnt816  
pnt817  
pnt818  
pnt819  
pnt820  
pnt821  
pnt822  
pnt823  
pnt824  
pnt825  
pnt826  
pnt827  
pnt828  
pnt829  
pnt830  
pnt831  
pnt832  
pnt833  
pnt834  
pnt835  
pnt836  
pnt837  
pnt838  
pnt839  
pnt840  
pnt841  
pnt842  
pnt843  
pnt844  
pnt845  
pnt846  
pnt847  
pnt848  
pnt849  
pnt850  
pnt851  
pnt852  
pnt853  
pnt854  
pnt855  
pnt856  
pnt857  
pnt858  
pnt859  
pnt860  
pnt861  
pnt862  
pnt863  
pnt864  
pnt865  
pnt866  
pnt867  
pnt868  
pnt869  
pnt870  
pnt871  
pnt872  
pnt873  
pnt874  
pnt875  
pnt876  
pnt877  
pnt878  
pnt879  
pnt880  
pnt881  
pnt882  
pnt883  
pnt884  
pnt885  
pnt886  
pnt887  
pnt888  
pnt889  
pnt890  
pnt891  
pnt892  
pnt893  
pnt894  
pnt895  
pnt896  
pnt897  
pnt898  
pnt899  
pnt900  
pnt901  
pnt902  
pnt903  
pnt904  
pnt905  
pnt906  
pnt907  
pnt908  
pnt909  
pnt910  
pnt911  
pnt912  
pnt913  
pnt914  
pnt915  
pnt916  
pnt917  
pnt918  
pnt919  
pnt920  
pnt921  
pnt922  
pnt923  
pnt924  
pnt925  
pnt926  
pnt927  
pnt928  
pnt929  
pnt930  
pnt931  
pnt932  
pnt933  
pnt934  
pnt935  
pnt936  
pnt937  
pnt938  
pnt939  
pnt940  
pnt941  
pnt942  
pnt943  
pnt944  
pnt945  
pnt946  
pnt947  
pnt948  
pnt949  
pnt950  
pnt951  
pnt952  
pnt953  
pnt954  
pnt955  
pnt956  
pnt957  
pnt958  
pnt959  
pnt960  
pnt961  
pnt962  
pnt963  
pnt964  
pnt965  
pnt966  
pnt967  
pnt968  
pnt969  
pnt970  
pnt971  
pnt972  
pnt973  
pnt974  
pnt975  
pnt976  
pnt977  
pnt978  
pnt979  
pnt980  
pnt981  
pnt982  
pnt983  
pnt984  
pnt985  
pnt986  
pnt987  
pnt988  
pnt989  
pnt990  
pnt991  
pnt992  
pnt993  
pnt994  
pnt995  
pnt996  
pnt997  
pnt998  
pnt999  
pnt1000

pnt7  
pnt8

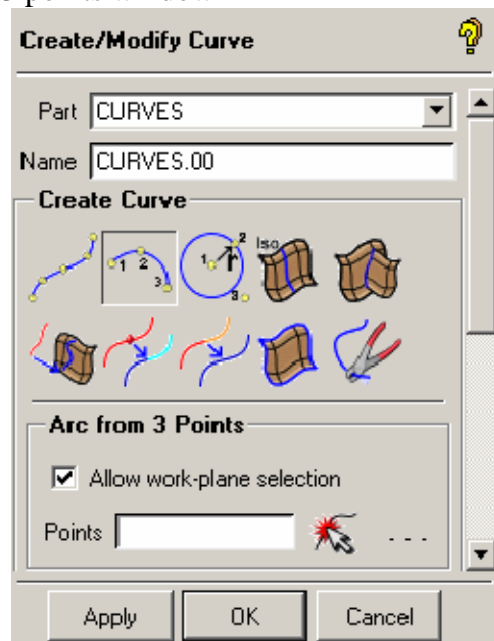



**TIP1:** Switch ON the **Points** in the left side *model Tree window*. To see the names of the points, use the **right mouse** and select **button Points > show Points Names** in the *model Tree window*. To turn Off **Points Names**, click button **Points > show Points Names** in the *model Tree window* again.

## b) Line Creation

**Geometry > Create/Modify Curve > Arc from 3 Points:** Select the  (Arc from 3 Points) to open the *From Points* window as shown in Figure D06.

Figure D06: Arc from 3 points window



Unclick on **Allow work-plane selection**. To select Points, click on  (Entity chooser icon) and then select **pnt0**, **pnt1** and **pnt2** with the left-mouse button. Press the middle mouse button to accept the points. The points name will appear in the select window. Enter the **Part** to **BLADES**, and **Name** to **BLADES.00**. Press **Apply** to create a curve.

Similarly make other lines by just selecting the points below, the curves name will then change by default.

BLADES.01: pnt4, pnt5 and pnt6

BLADES.02: pnt0, pnt8 and pnt6

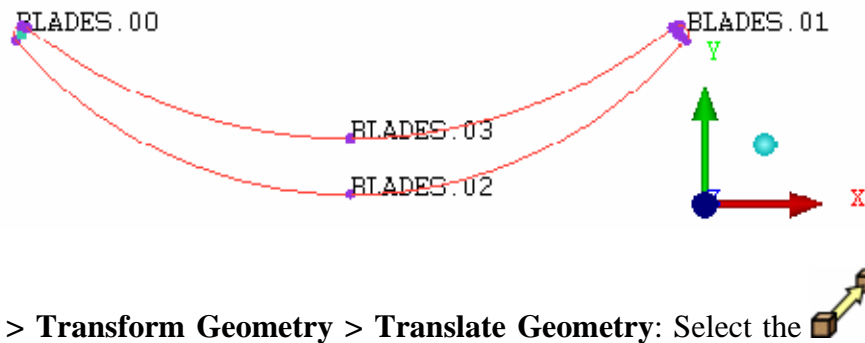
BLADES.03: pnt2, pnt7 and pnt4

**TIP2:** At any time during the selection process, pressing F9 will allow the user to zoom, rotate or pan to get a better view. Toggle F9 again will return the user to selection mode.



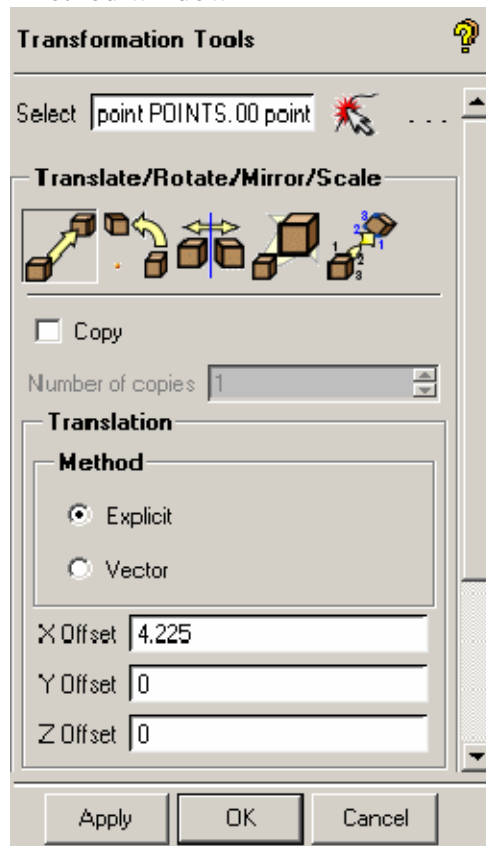
**TIP3:** To see the names of curves, use the right mouse and select **button Curves > show Curves Names** in the *model Tree* window. It is shown in Figure A07.

Figure D07: Geometry after blade curves generated







**Geometry > Transform Geometry > Translate Geometry:** Select the  (Translate Geometry) to open the *Select* window as shown in Figure D08.

Figure D08: Translation method window



Switch ON **Points** and **Curves** in the left side *model Tree* window.



To select **Rotating objects**, click on  and Click on  from **Select Geometry** after switching off  and . Under **Rotation**, set on **Axis** to **Z**, and set **Angle** to **32.605**. Under **Center of Rotation**, set **Center Point** to **Specified** and **Location** to (4.225 0 0). Press **Apply** to rotate points and curves.

### c) Blade Surface Creation




**Geometry > Create/Modify Surface > From Curves:** Select the  (From Curves) to open the *Select Curves* window as shown in Figure D11.

Figure D11: Select curves window



To select Curves, click on  and select BLADES.00, BLADES.01, BLADES.02 and BLADES.03 with the left mouse button. Press the middle mouse button to accept the curves. The curves name will appear in the select window. Set **Tolerance** to 0, and enter the **Part** as **BLADES**, and **Name** as **BLADES.00**. Press **Apply** to create the surface.

**TIP4:** To regroup **Parts**, double-click **Parts** in the left side *model Tree window* with the left-mouse button, the **Create Part** window appear as shown in Figure A12. Switch OFF **Curves** and **Surfaces** in the left side *model Tree window*. Set **Parts** to **POINTS** and click on  from **Select Geometry**. The imported points are joining into **POINTS** by pressing **Apply**.

**TIP5:** To delete **empty Parts**, click **Parts > Delete Empty Parts** in the *model Tree window*.






**Geometry > Transform Geometry > Rotate Geometry:** Switch ON **Curves** and **Surfaces** in the left side *model Tree window*. Switch OFF **Points**. To select rotating objects, click on  and select  from **Select Geometry** after switching ,  and  on, then Check **Copy** and Set **Number of copies** to **29**. Under **Rotation**, set **Axis** to **Z**. Set **Angle** to **12** and **Center of Point** to **Origin**. Press **Apply** to copy curves and surfaces. The result is shown in Figure D13.

Figure D12: Create part window

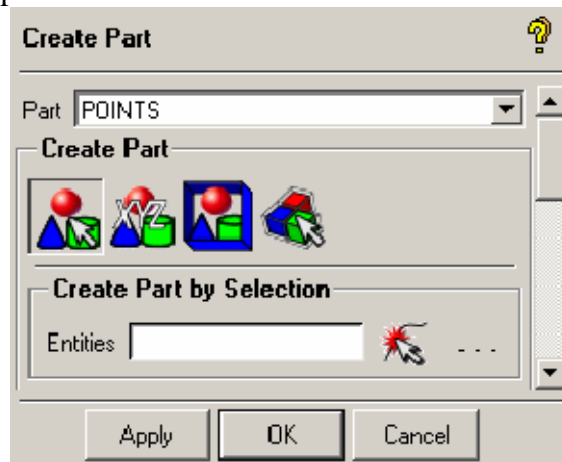


Figure D13: Geometry after blade surface generated

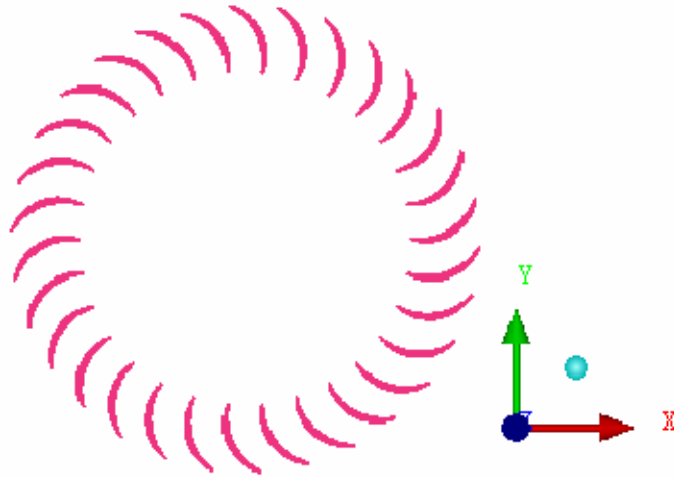
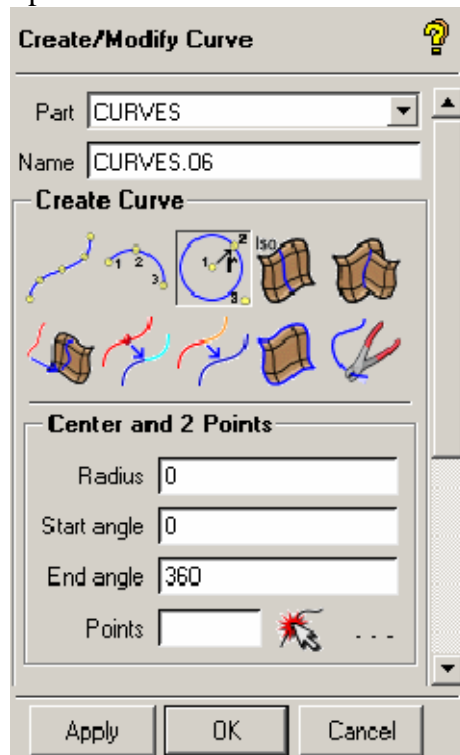



Figure D14: Center and 2 points window



**d) Surface around blade Creation**

**Geometry > Create/Modify Curves > Circle or arc from Center point and 2 points**

**on plane:** Select the  (Circle or arc from Center point and 2 points on plane) to open *Center and 2 points* window as shown in Figure D14.


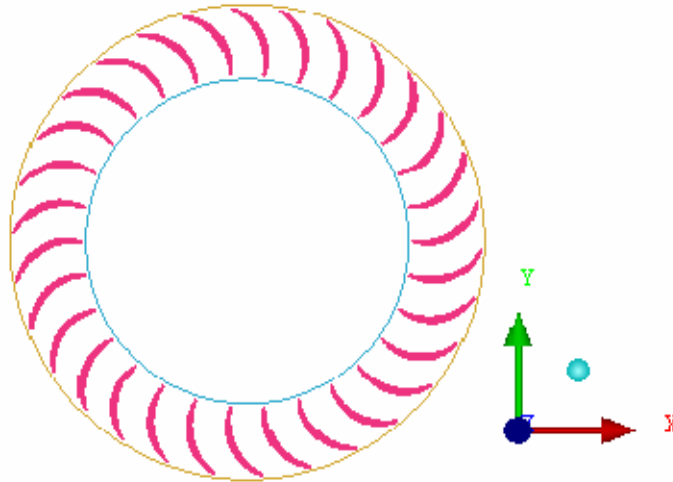


To select Center Point, click on  and select point with coordinate ( 0 0 0) with the left mouse button. Press the middle mouse button to accept the point. If there is no such point, then create one. The point name will appear in the select window. Enter the **Part** to **INNERCIRCLE**, and set **Name** to **INNERCIRCLE.00**. Set **Radius** to **4.1**, and Press **Apply** to create the circle. Similarly make other circle with **Radius equal to 6**. Enter the **Part** to **OUTERCIRCLE**, and set **Name** to **OUTERCIRCLE.00**. The result is shown in Figure D15.

Figure D15: Geometry after two circles created



**Geometry > Create Points > Parameter along a curve:** Select the  (Parameter along a Curve) to open *Parameter along a Curve* window as shown in Figure D16.

To select Curve, click on  and select BLADES.00 with the left mouse button. Press the middle mouse button to accept the curve. Enter the **Part** as **POINTS**, and set **Name** by default. Set **Curve parameters** to **0.5**. Press **Apply** to create the point. Similarly make center points for other three edges of a blade.

Repeat the same steps to create center points for other blades. The final result is shown in Figure D17.

Figure D16: Parameter along a curve window

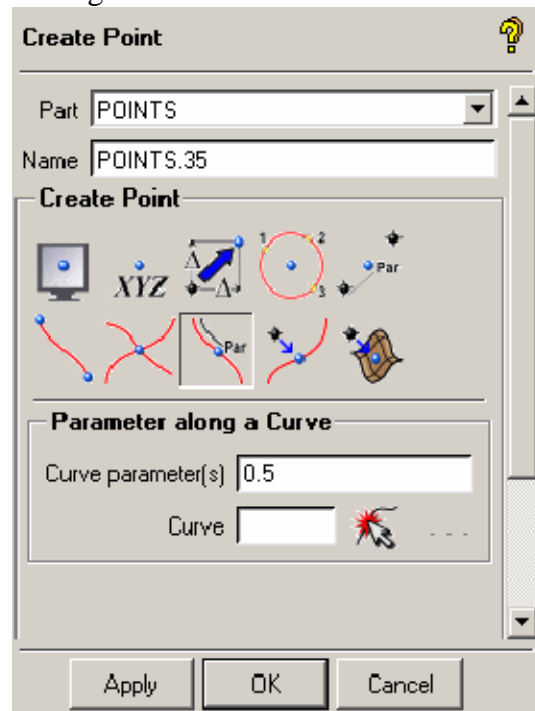
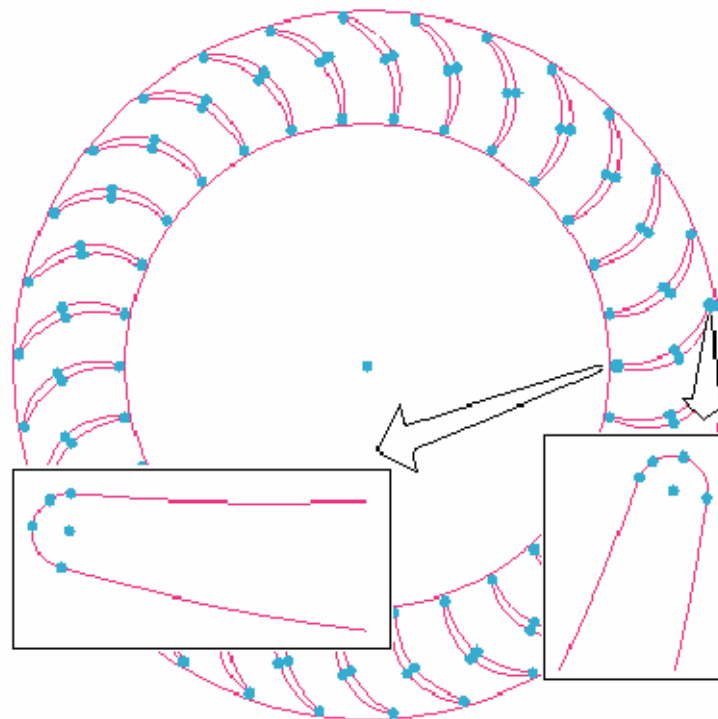


Figure D17: Geometry after center point at each edge of blades created




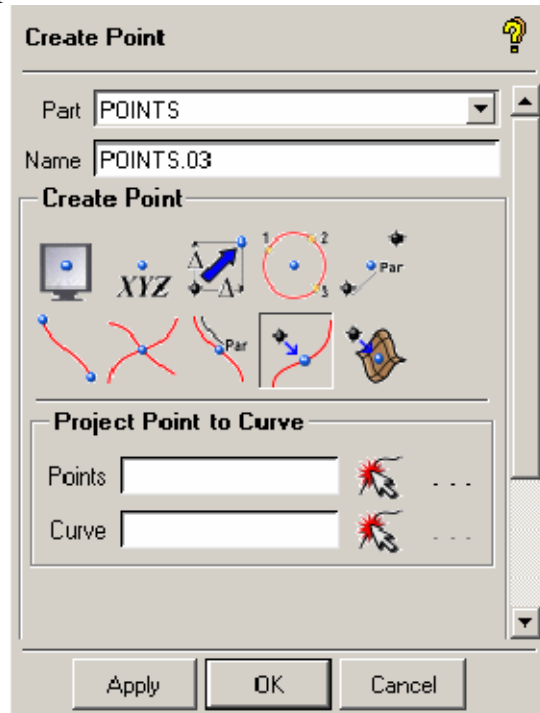
**Geometry > Create Points > Project point to curve:** Select the  (Project point to Curve) to open *Project Point to Curve* window as shown in Figure D18.

Figure D18: Project point to curve



Project points created at leading edge and trailing edge previously to INNERCIRCLE and OUTERCIRCLE respectively. Enter the **Part** to **POINTS**, and set **Name** by default.

**Geometry > Create/Modify Curve > Segment curve:** Select the  (Segment Curve) to open *Segment Curve* window as shown in Figure D19.

To Segment INNERCIRCLE by selecting four points with almost even distance. Enter the **Part** as **INNERCIRCLE**, and set **Name** by default.

**Geometry > Create/Modify Surface > From Curves:** To make inner circle surface by selecting four segmented curves with **tolerance** equal to **0.000001**. Enter the **Part** as **SYM1**, and set **Name** by default.

**Geometry > Create/Modify Curve > Segment curve:** Continue to split INNERCIRCLE and OUTERCIRCLE by other points on them respectively. Enter the **Part** to **INNERCIRCLE** for INNERCIRCLE and **OUTERCIRCLE** for OUTERCIRCLE. Set **Name** by default for both cases.

In the same manner, split curves BLADES.00 and BLADES.01 with the points at the center respectively. Repeat the same steps for other blades. Enter the **Part** as **BLADES**, and set **Name** by default.




**Geometry > Create/Modify Curve > From Points:** Select the  (From Points) to open *From Points* window as shown in Figure D20.

To create curves between blades and circles by joining two points and set **Tolerance** to **0**, enter the **Part** to **CURVES**, and set **Name** by default. The result is shown in Figure D21.


**Geometry > Create Points > Parameter along a curve:** Select each segment curve on INNERCIRCLE and OUTERCIRCLE. Enter the **Part** as **POINTS**, and set **Name** by default. Set **Curve parameters** to **0.5**. Press **Apply** to create the points.

**Geometry > Create/Modify Curve > Segment curve:** Segment circles again by the points just created.

**Geometry > Create Point > Parameter Along a Vector:** Select the  (Parameter along a Vector) to open *Parameter along a Vector* window as shown in Figure D22.

To select 2 points, click on  and select points at the center of the pressure edge and suction edge with the left mouse button. Press the middle mouse button to accept the point. Set **Parameter(s)** to **0.5**, Enter the **Part** as **POINTS**, and set **Name** by default. Press **Apply** to create points.

**Geometry > Create/Modify Curve > Arc from 3 Points:** Create Curves between blades as shown in Figure D23, Enter the **Part** as **CURVES**, and **Name** by default.

**Geometry > Create/Modify Surface > From Curves:** To select Curves, click on  and select curves shown in Figure D24 with the left mouse button. Press the middle mouse button to accept the curves. Enter the **Part** as **SYM1**, and **Name** by default, Set **Tolerance** to **0.001**. Press **Apply** to create the surface.

Follow the same steps and Repeat **59** times to create other surfaces around blades, be sure that curves can enclose an area.

Figure D19: Segment curve windowmove this to the next page – check all the way through

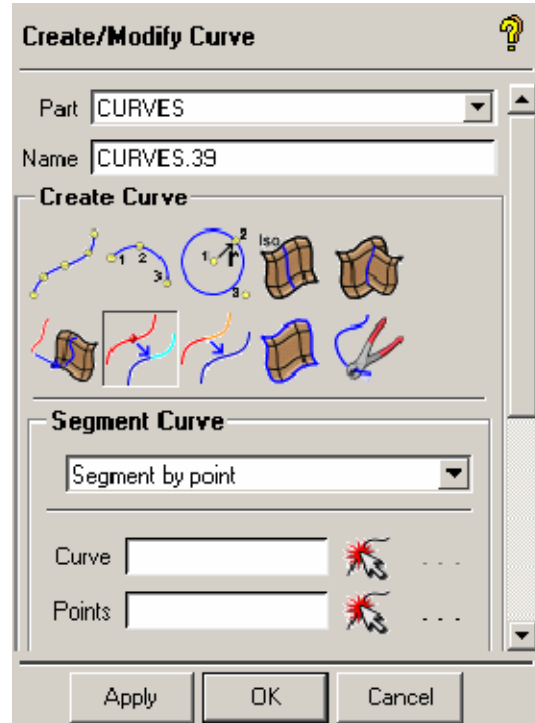


Figure D20: Form points window

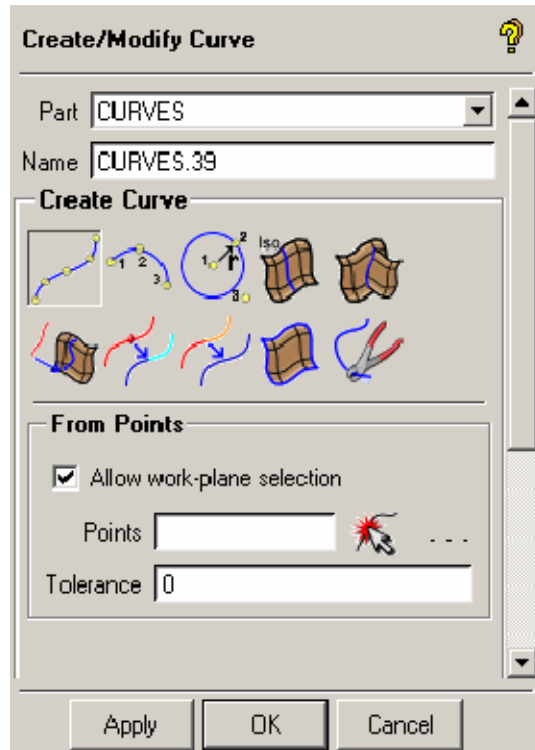


Figure D21: Geometry after assistant curves created

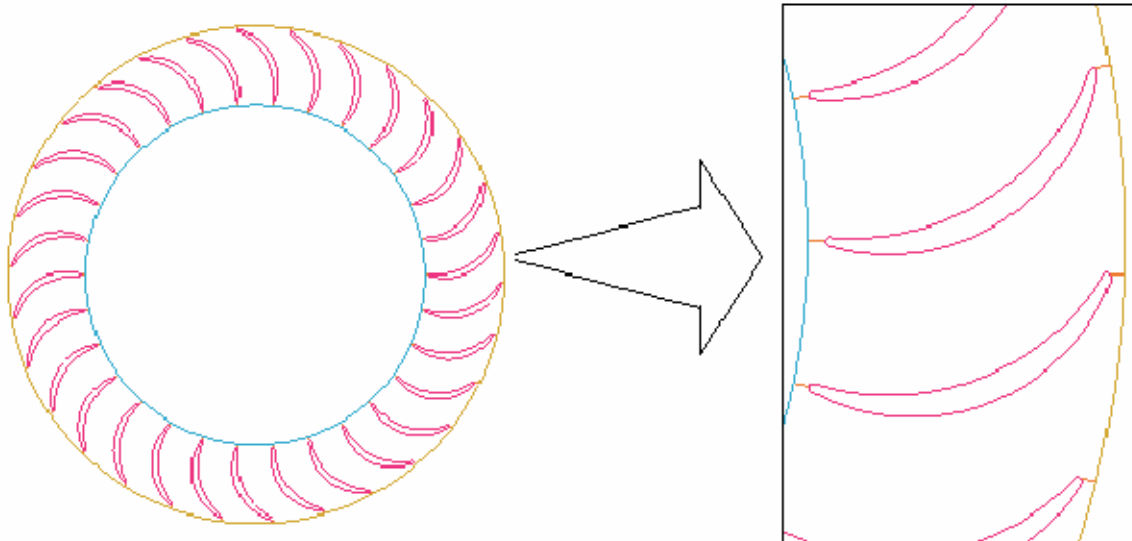


Figure D22: Parameter along a vector window

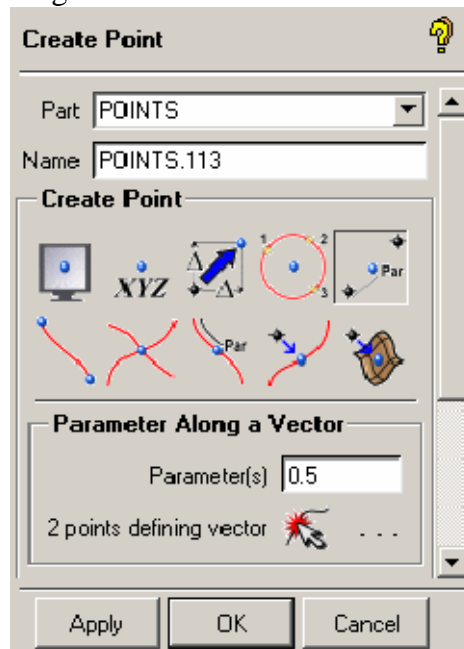


Figure D23: Geometry after assistant curves created

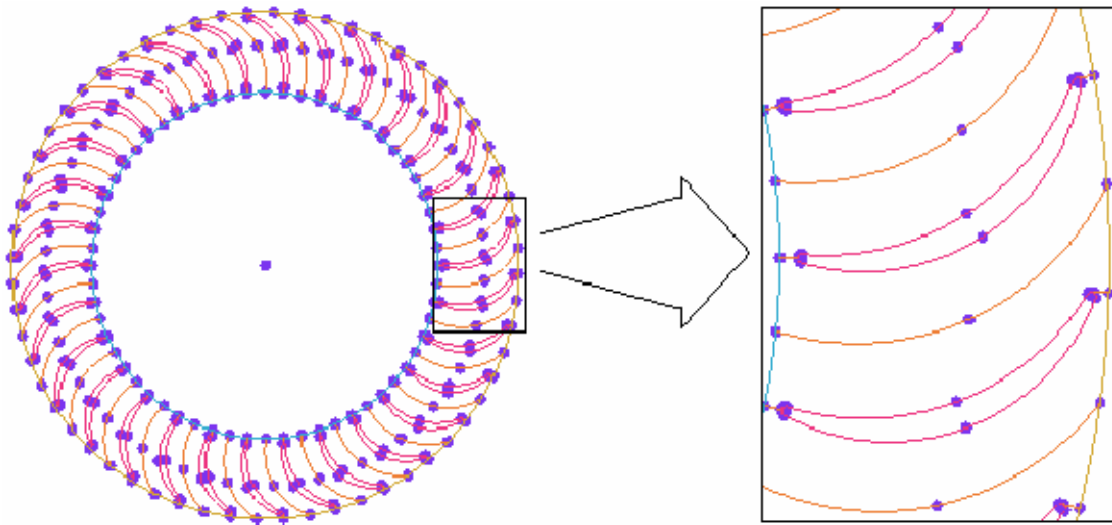
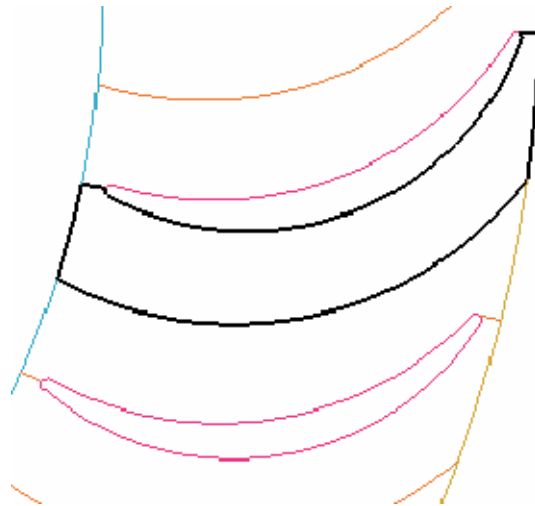


Figure D24: The selection of surface creation around the blade

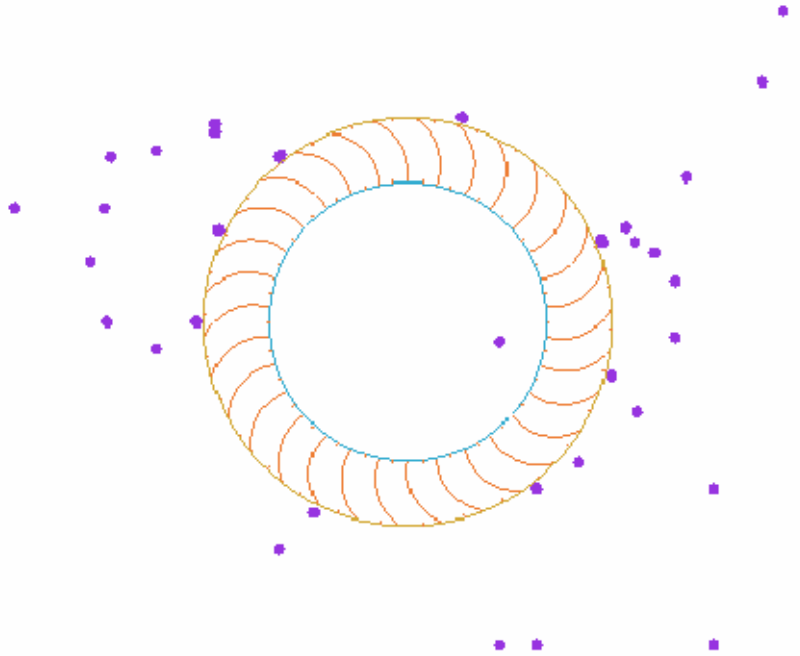


## 2.2. Cavities, Inlet, Outlet Section Generation


### a) Points Creation

Select **File > Import Geometry > Formatted point data** from the Main menu and choose **Cavities.txt** as shown in Figure D03. Uncheck **Import Curves**, **Import Surfaces** and **Plot3D Format**, then Set **Approximation Tolerance** to **0**. To accept the points by click **Apply**, then it is read and shown in Figure D25.

Figure D25: Geometry after cavities points imported




#### b) Line Creation

**Geometry > Create/Modify Curve > From Points:** To select Points, click on  and then select **pnt9, pnt10, pnt11, pnt12, pnt13, pnt14, pnt15 and pnt16** with the left mouse button. Press the middle mouse button to accept the points. The points name will appear in the select window. Set **Tolerance** to **0**, Enter the **Part** to **WALL**, and **Name** to **WALL.00**. Press **Apply** to create the curve for high pressure cavity.

Similarly make other curves by selecting the following points, Leave the **Part** as **WALL** and **Name** by default.

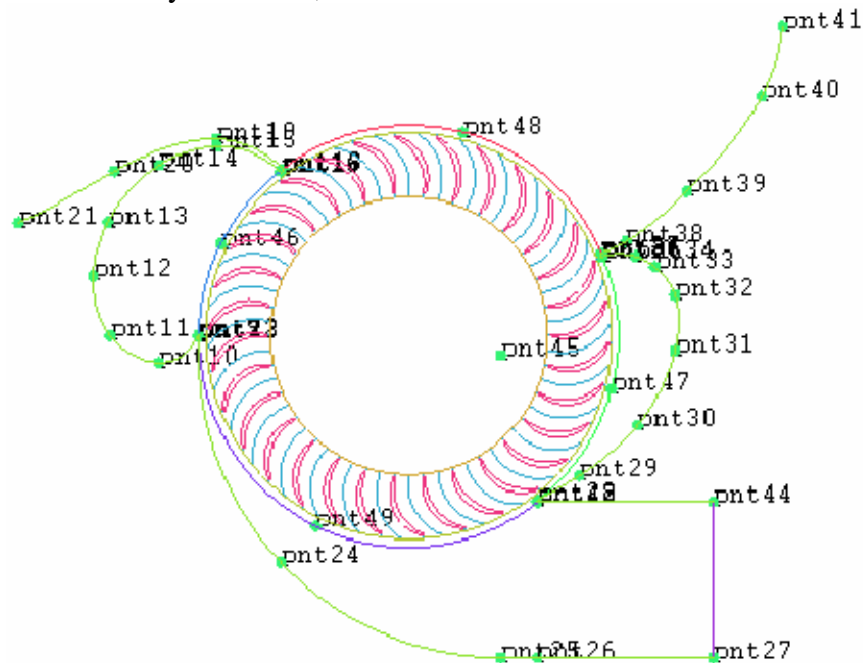
- WALL.01: pnt28, pnt29, pnt30, pnt31, pnt32, pnt33, pnt34 and pnt35 (Low Pressure Cavity)
- WALL.02: pnt18, pnt19, pnt20 and pnt21 (Inlet Section)
- WALL.03: pnt37, pnt38, pnt39, pnt40 and pnt41 (Inlet Section)
- WALL.04: pnt25, pnt26 and pnt.27 (Set **Tolerance** to **0.00001**, others are 0) (Outlet Section)
- WALL.05: pnt43 and pnt44 (Outlet Section)
- OUTLET.00: pnt27 and pnt44 (Set **Part** as **OUTLET** and **Name** as **OUTLET.00**) (Outlet Section)

**Geometry > Create/Modify Curve > Arc from 3 Points:** Click on  and then select **pnt16**, **pnt17** and **pnt18** with the left mouse button. Press the middle mouse button to accept the points. The points name will appear in the select window. Enter the **Part** as WALL and **Name** as WALL.06. Press **Apply** to create the curve.

Similarly make other lines by selecting the group of points shown below separately:


- WALL.07: pnt9, pnt22 and pnt23
- WALL.08: pnt23, pnt24 and pnt25
- WALL.09: pnt28, pnt42 and pnt43
- WALL.10: pnt35, pnt36 and pnt37
- INT0.00: pnt23, pnt49 and pnt43 (Set **Part** as INT0 and **Name** as INT0.00)
- INT1.00: pnt28, pnt47 and pnt35 (Set **Part** as INT1 and **Name** as INT1.00)
- INT2.00: pnt37, pnt48 and pnt18 (Set **Part** as INT2 and **Name** as INT2.00)
- INT3.00: pnt22, pnt46 and pnt16 (Set **Part** as INT3 and **Name** as INT3.00)


Figure D26: Geometry after wall, outlet and interface created




Since Inlet section is not complete, it is needed to continue the curve generation:

**Geometry > Create/Modify Curve > Segment curve:** Split curve WALL.02 by point pnt20.

**Geometry > Delete Any Entity:** Select the  (Delete Any Entity) from the main toolbar. Delete the curve between points: pnt20 and pnt21.

**Geometry > Create/Modify Curve > Modify Curves:** Select the  (Modify Curves) to open *Modify Curves* window as shown in Figure D27.

To select Curve, click on  and then select previous segment curve with the left mouse button. Press the middle mouse button to accept the curve. In the same manner, set **Location** to pnt21. Enter the **Part** as **WALL**, and **Name** by default. Press **Apply** to create the curve.

Continue to extend the curve, under **End point selection**, click **Length** and the window is shown in Figure D28, Set **Length** to **25**. Under **Extend as**, Click on **Line**. Enter the **Part** as **WALL**, and **Name** by default. Press **Apply** to extend the curve.

Similarly extend a curve WALL.03 by setting **Length** to **25**, and extend as a **Line**. The final curves are shown in Figure D29.

Figure D27: Modify curves by selecting end point window

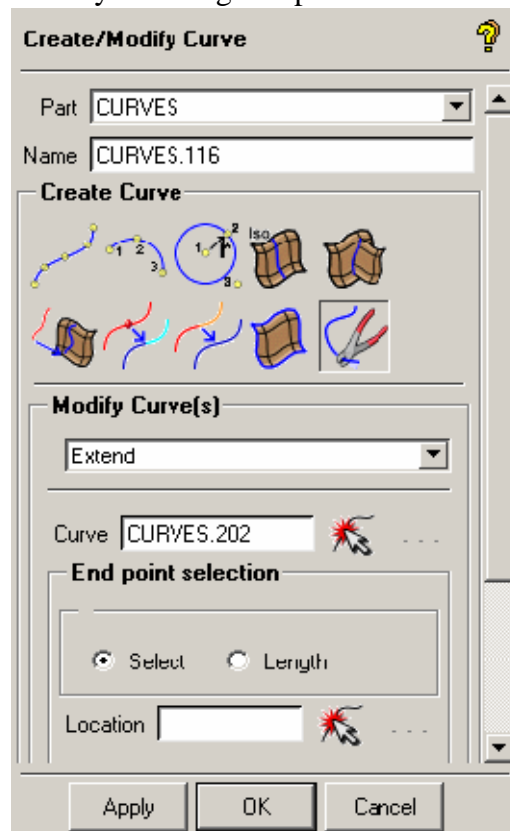


Figure D28: Modify curves by extending length window

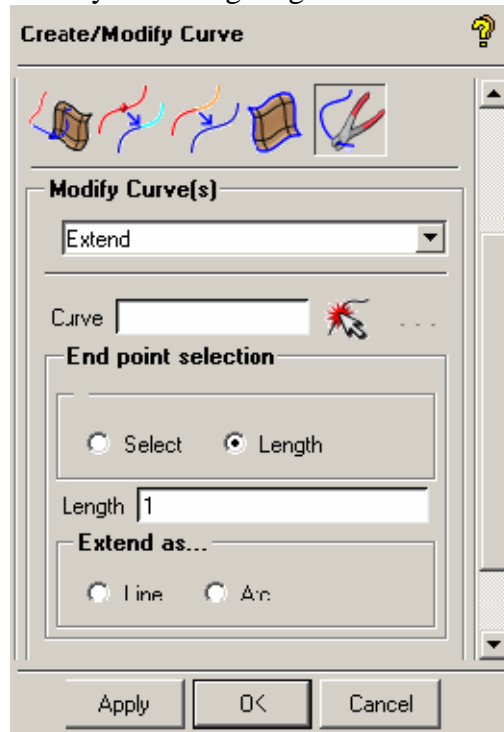
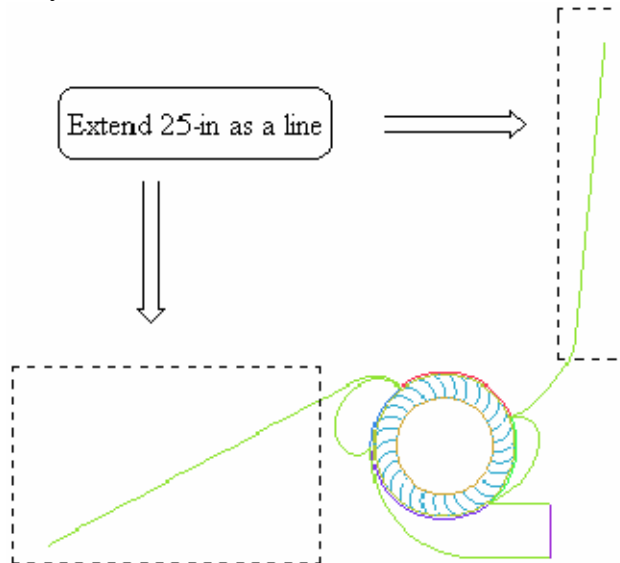


Figure D29: Geometry after curves extended at inlet section



**Geometry > Create/Modify Curves > Circle or arc from Center point and 2 points on plane:** To create a circle with radius **25** along center point (0 0 0). Enter the **Part** as **INLET**, and set **Name** to **INLET.00**. Press **Apply** to create a circle.




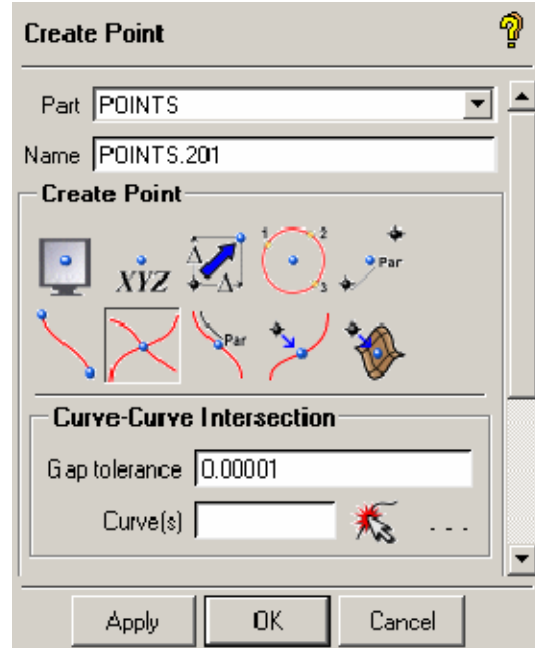
**Geometry > Create Point > Curve-Curve Intersection:** Select the  (Curve- Curve Intersection) to open the *Curve-Curve Intersection* window as shown in Figure D30.

Figure D30: Curve-Curve intersection window




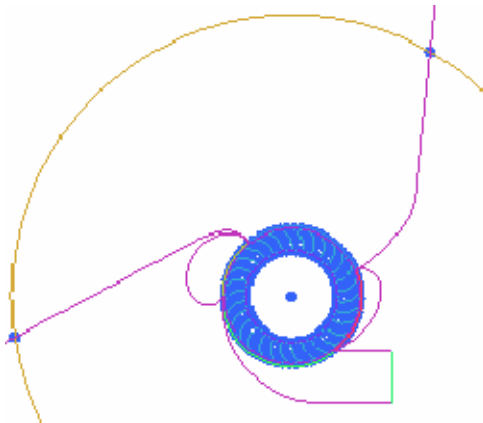
To select Curve, click on  and then select **INLET.00** and **extended Curves** with the left mouse button. Press the middle mouse button to accept the curve. Set **Gap tolerance** to 0.000001, Enter the **Part** as **POINTS**, and **Name** by default. Press **Apply** to create point as shown in Figure D31.

Figure D31: Geometry after circle with radius 25 inch and intersection points created



**Geometry > Create/Modify Curve > Segment curve:** Spilt Curve INLET.00 and extended curves WALL.03 and WALL.04 with two intersection points.


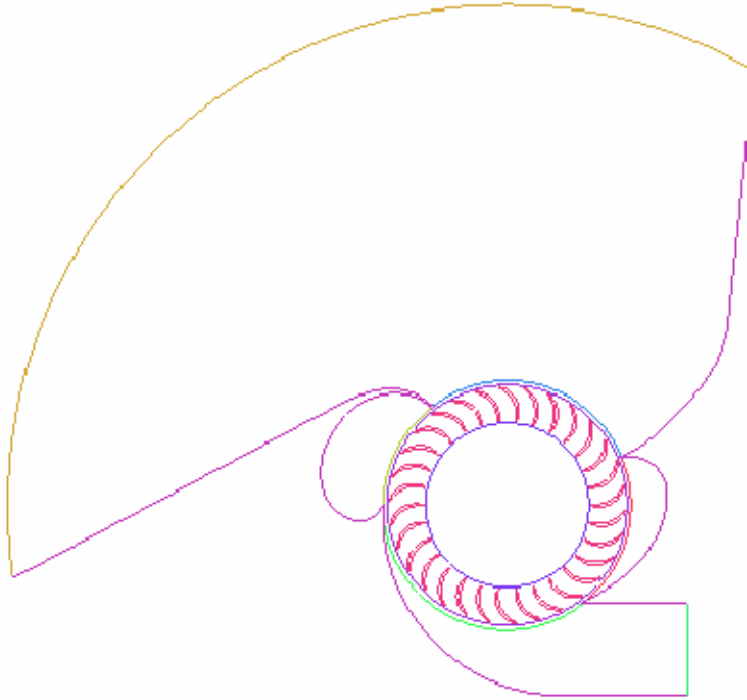
**Geometry > Delete Any Entity:** Select the  from the main toolbar. Delete the curves and the final geometry is shown in Figure D32.

Figure D32: Geometry after inlet section created

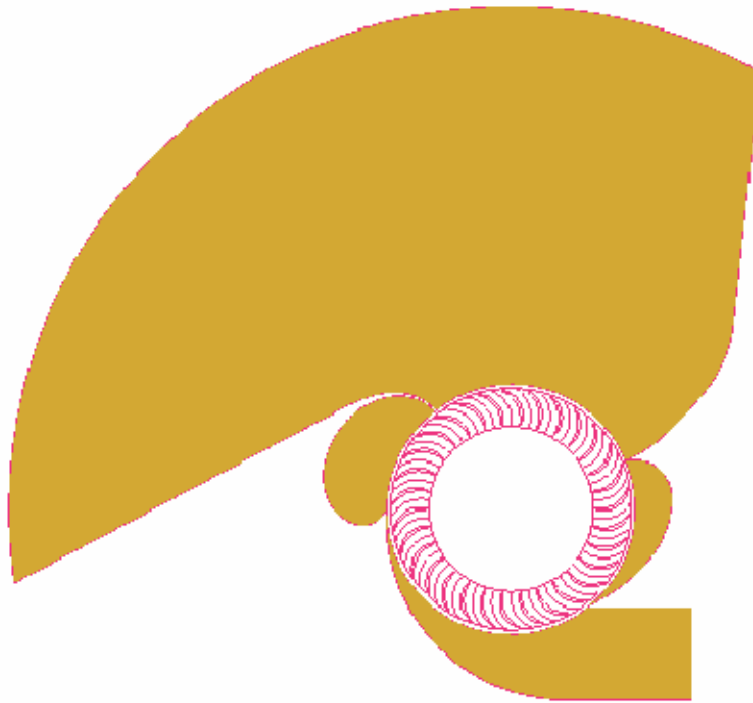


**TIP6:** To see the better color distribution, use the **right mouse** and select **button Part > Good Colors** in the *model Tree window*.

**c) Surface for Inlet Section, Outlet Section, High Pressure Cavity and Low Pressure Cavity generation**

**Geometry > Create/Modify Surface > From Curves:** Select curves around with the left mouse button. Press the middle mouse button to accept the curves. No matter there is four curves or not, as long as the area is closed, the surface will be generated. For Inlet Section, High Pressure and Low Pressure Cavity, set **Tolerance** to 0. For Outlet Section, set **Tolerance** to 0.00001. And it is shown in Figure D33.

Figure D33: Geometry after surface at inlet section, outlet section, high pressure cavity and low pressure cavity created



In order to generate the surface between OUTERCIRCLE.00 (A circle with radius 6-in) and 4 sections (inlet section, outlet section, high pressure cavity and low pressure cavity), there are more assistant points and curves required.

**Geometry > Create Points > Project point to curve:** Project points on OUTERCIRCLE to its outer curves corresponding to inlet section, outlet section, high pressure cavity and low pressure cavity. Enter the **Part** as **POINTS**, and leave **Name** by default. Similarly create four points on curve OUTERCIRCLE by project points: pnt17, pnt23, pnt42, and pnt36.

**Geometry > Create/Modify Curve > Segment curve:** Split the curves by the points projected previously.

**Geometry > Create/Modify Curve > From Points:** Create curve between the gaps by two corresponding points created previously. Set **Tolerance** to **0**, enter the **Part** as **CURVES** and set **Name** by default. However, there are four curves excluded and shown in Figure D34.

- |         |  |
|---------|--|
| INT5.00 | pnt17 and its projected point on OUTERCIRCLE (Set <b>Part</b> as <b>INT5</b> and <b>Name</b> as <b>INT5.00</b> ) |
| INT6.00 | pnt23 and its projected point on OUTERCIRCLE (Set <b>Part</b> as <b>INT6</b> and <b>Name</b> as <b>INT6.00</b> ) |

INT7.00      pnt42 and its projected point on OUTERCIRCLE (Set **Part** as **INT7**  
and **Name** as **INT7.00**)

INT8.00      pnt36 and its projected point on OUTERCIRCLE (Set **Part** as **INT8**  
and **Name** as **INT8.00**)

PS: The part INT4 is kept for an assistant curve for future mesh generation and it will be created at next section.

**Geometry > Create/Modify Surface > From Curves:** Select curves, basically four curves create a surface, but even there are no four curves, a surface will be created due to enclosed area. Follow the steps indicated and Set **Tolerance** to **0.001**. Enter the **Part** as **SYM1**, and leave **Name** by default.

If there are more points or curves needed, follow the steps above to create them. The result is shown in Figure D34 and four particular areas are indicated.

#### **d) Assistant Curves for Future Mesh Generation**

There is a curve which will be used for future Mesh Generation, and it is created by following steps:

**Geometry > Create Points > Parameter along a curve:** Set **Curve Parameter(s)** to 0.8 and select curve shown at the right of Figure D35. Similarly make a point as shown in a middle of Figure D35. Enter the **Part** as **POINTS**, and set **Name** by default.

**Geometry > Create/Modify Curve > Segment curve:** Split the curve by point created previously.

**Geometry > Create/Modify Curve > Arc from 3 Points:** Click on three points (2 points are created previously, other one is pnt42) and make an arc shown in Figure D35. Enter the **Part** as **INT4**, and set **Name** to **INT4.00**.

Figure D34: Geometry after surface between outer circles generated

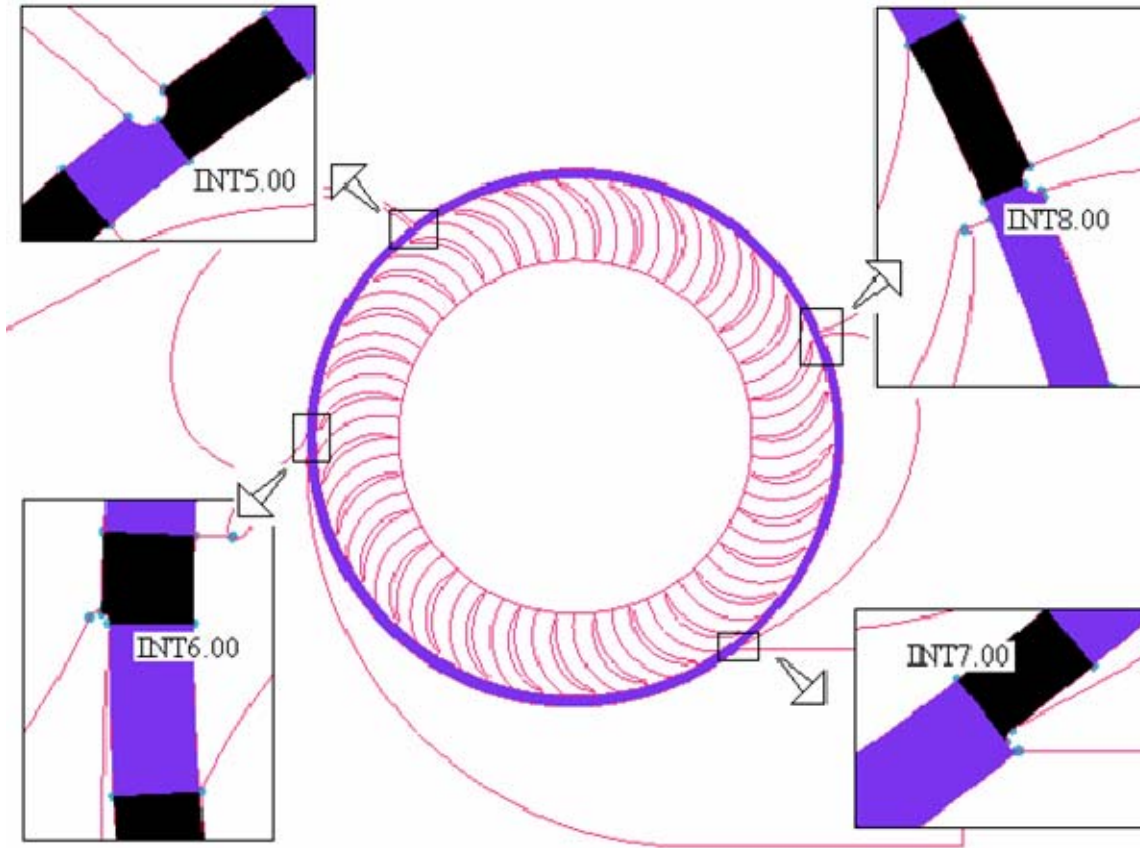
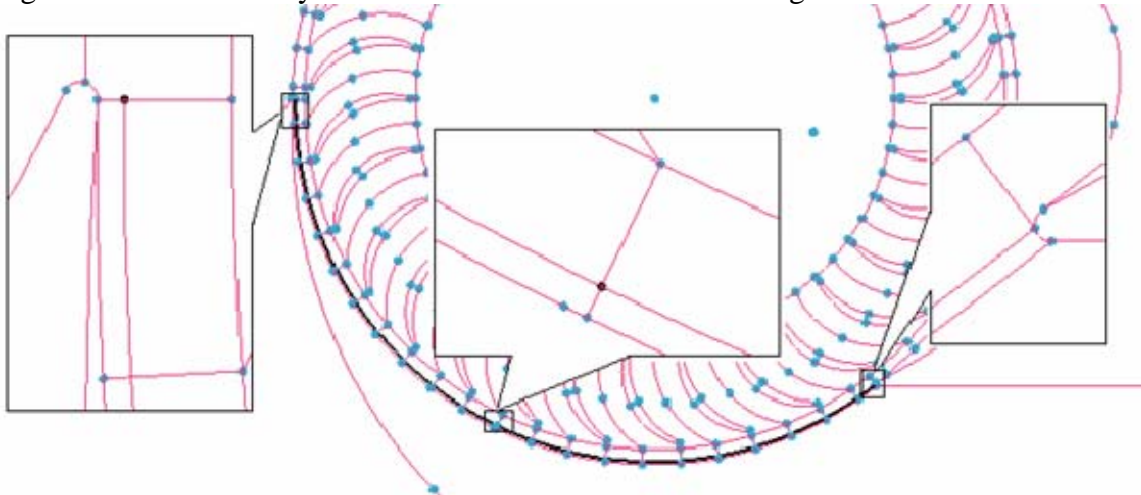


Figure D35: Geometry after assistant curve for future mesh generation



### 3. 3-D Model Generation

#### a) Extrude 0.1 inch along Z-axis

**Geometry > Create Point > Explicit Coordinates:** Create a point (0 0 0.1). Enter the **Part** as **POINTS**, and leave **Name** by default.


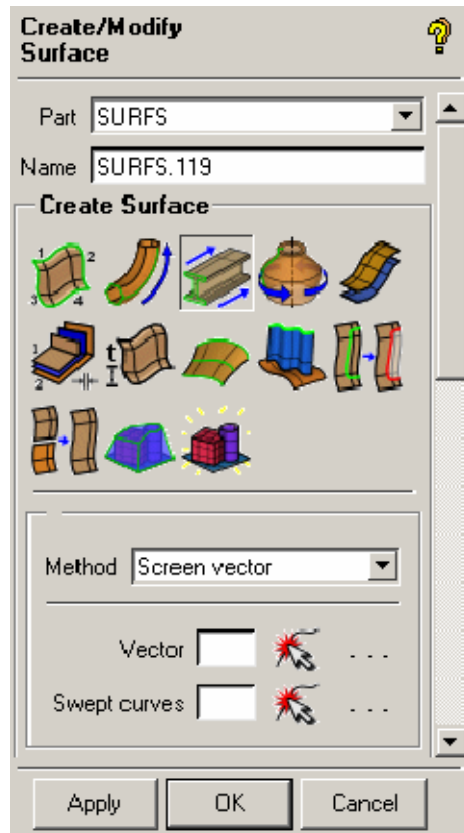

**Geometry > Create/Modify Surface > Sweep Surface:** Select the  (Sweep Surface) to open the *Select* window as shown in Figure D36.

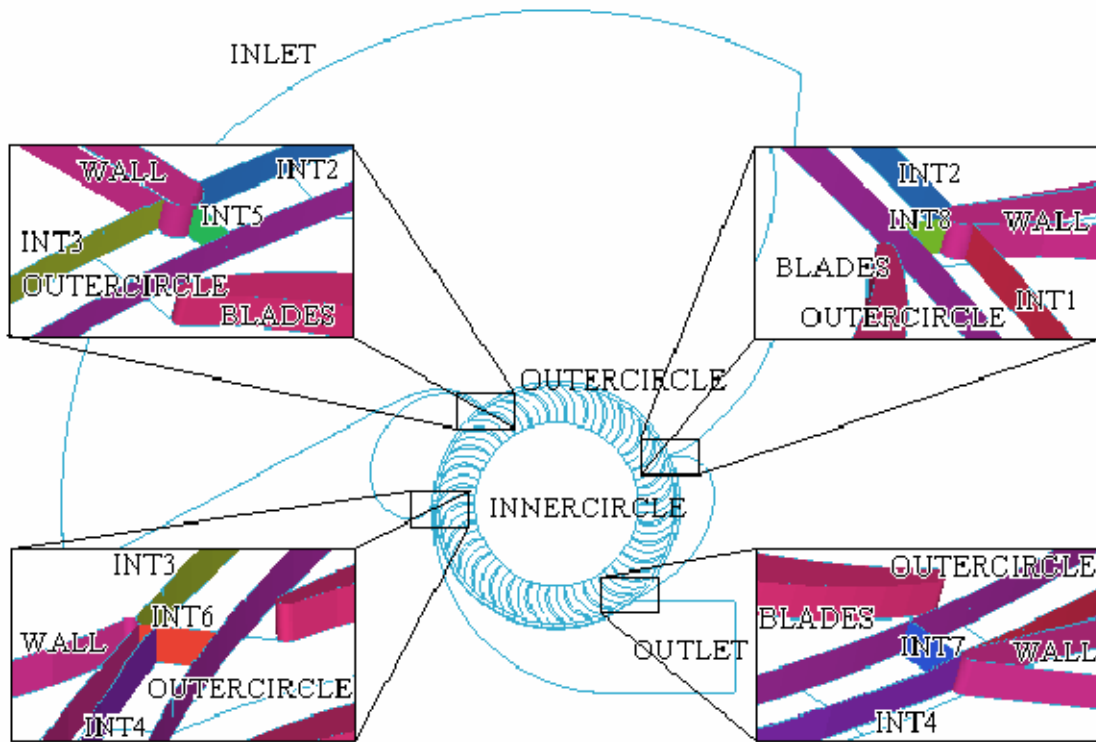
Figure D36: Sweep surface window



Set **Method** to **Screen Vector**, and set **Vector** to two points (0 0 0) and (0 0 0.1). Switch ON **Curves** in the left side model Tree window. Switch OFF the **Part** of **CURVES** and **INT0** in the left side model Tree window. Enter **Part** as **SURFS** and **Name** as **SURFS.00**. After clicking on **Entity chooser icon**, select  from **Select Geometry**. Then press **Apply**, geometry is created.

Regroup each Part by following the TIP4 mentioned previously, and let all curves to the **Part** of **CURVES**, all points to the **Part** of **POINTS**. Others are indicated and shown in Figure D37.

Figure D37: Geometry after extruded



**Geometry > Transform Geometry > Translate Geometry:** Translate **SYM1**, including curves and surfaces, and copy along Z-axis about 0.1-inch by selecting **Explicit Method**.

Regroup the translated surface by following the TIP4 mentioned previously, and Let its **Part** as **SYM2**.

Similarly translate blade surfaces about 0.1-inch along Z-axis to enclose the blades.

#### b) Body Creation


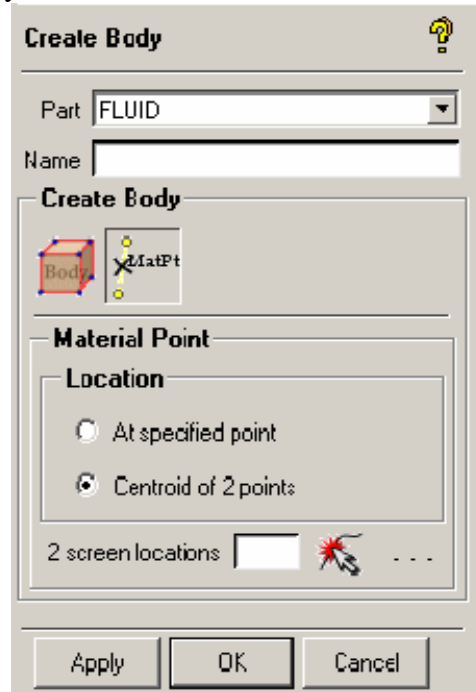
**Geometry > Create Body > Material Point > Centroid of 2 points:** Select the  (Create Body) to open create body window as shown in Figure D38. Enter **Part** name **FLUID** and select two point (0 0 0) and (0 0 0.1) with the left mouse button. Press the middle to accept the selection. Press **Apply** to create the material point.

Figure D38: Create body window



#### 4. Mesh Generation

##### 4.1. Inner Circle Mesh

###### a) Starting Blacking

**Blocking > Create Block**  **> Initialize block** : It will open the *Create Block* window as shown in Figure D39.

Under **Initialize Block**, set **Type** to **3D**. And select two inner-circle surfaces on planes **SYM1** and **SYM2**. Name **Part** as **FLUID** and press **Apply** to create block.



**Blocking > Associate**  **> Associate Vertex** : It will open the window as shown in Figure D40. Toggle **ON** Geometry > Points and Blocking > vertices > Numbers from tree widget. Under **Entity**, select **Point** and associate vertex to point as shown in Figure D41.



Figure D39: Create block window

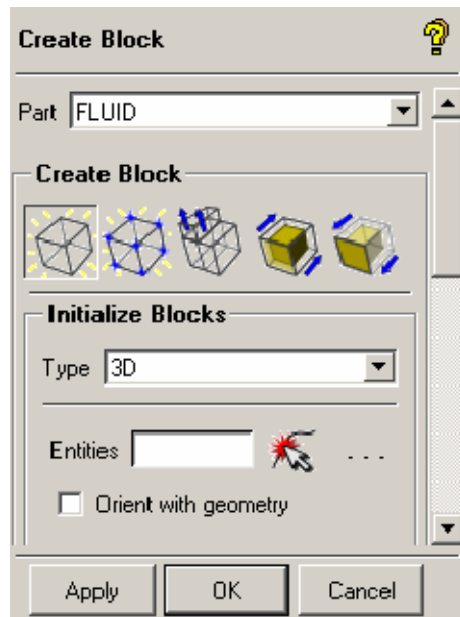


Figure D40: Associate vertex window

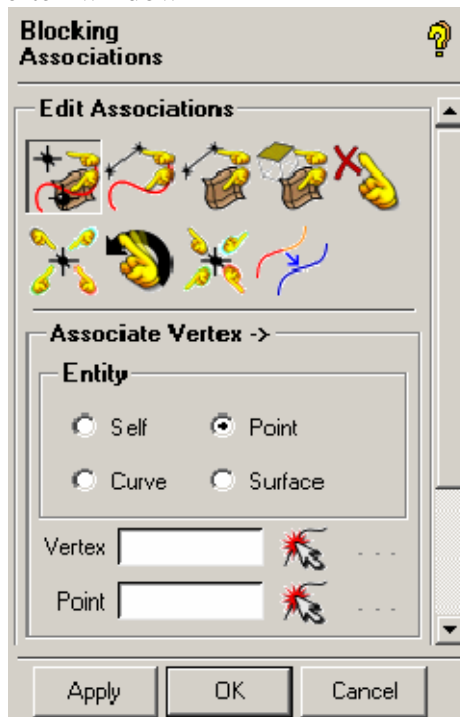
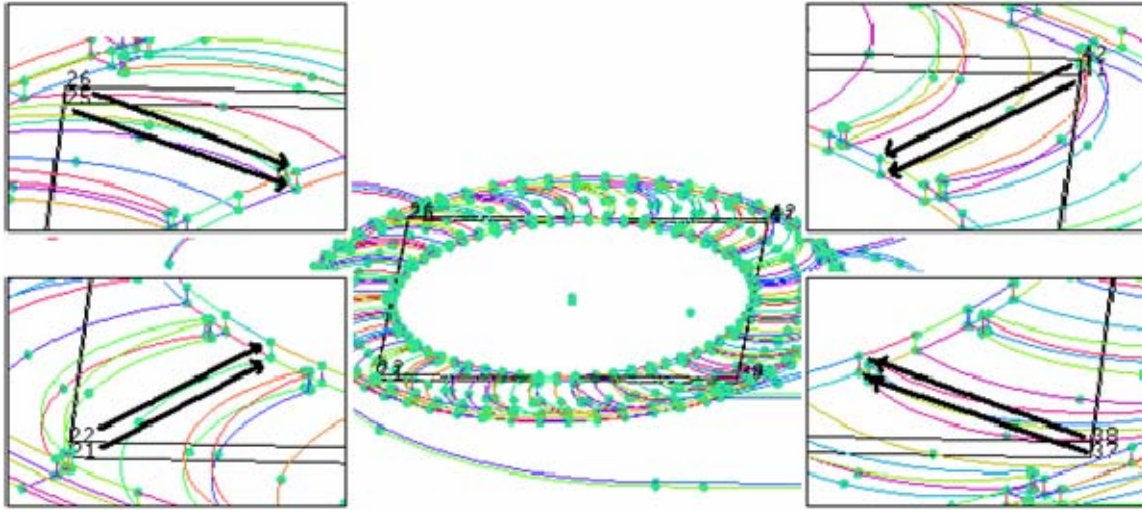


Figure D41: Moving vertices





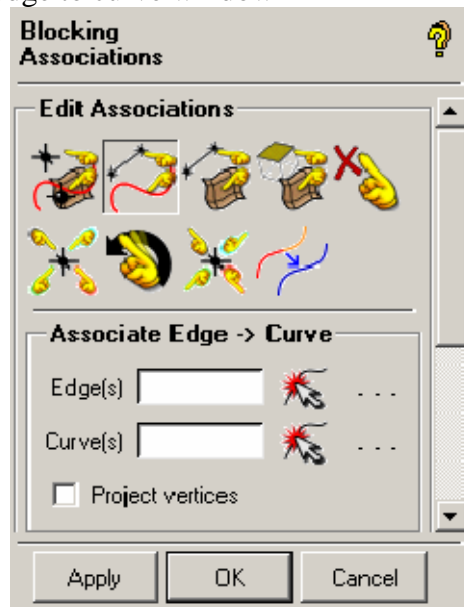
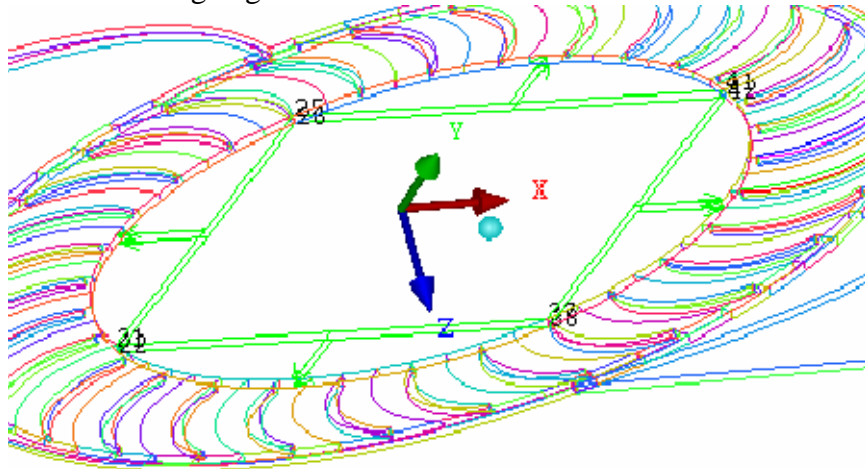
**Blocking > Associate**  **> Associate Edge to Curve**  : It will open the window as shown in Figure D42 and Associate edges to curves as shown in Figure D43.

Figure D42: Associate edge to curve window



**TIP:** To make sure the association is correct by selecting **Edges > Show Association** from the *Tree Widget* and switch on Curve

Figure D43: Associating edges to curves





**Blocking > Split Block**  **> Ogrid Block**  : It will open the *Ogrid block* window as shown in Figure D44. Select **Face(s)** as shown in Figure D45, Set **Offset** to 1. Press **Apply** to create O-grid block.

Figure D44: O-grid block window

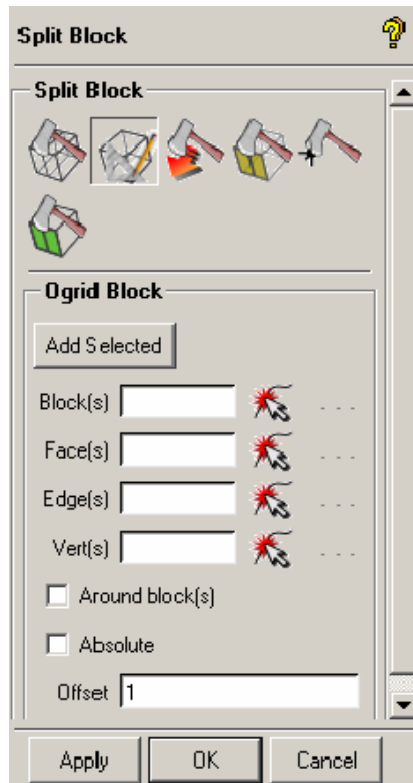
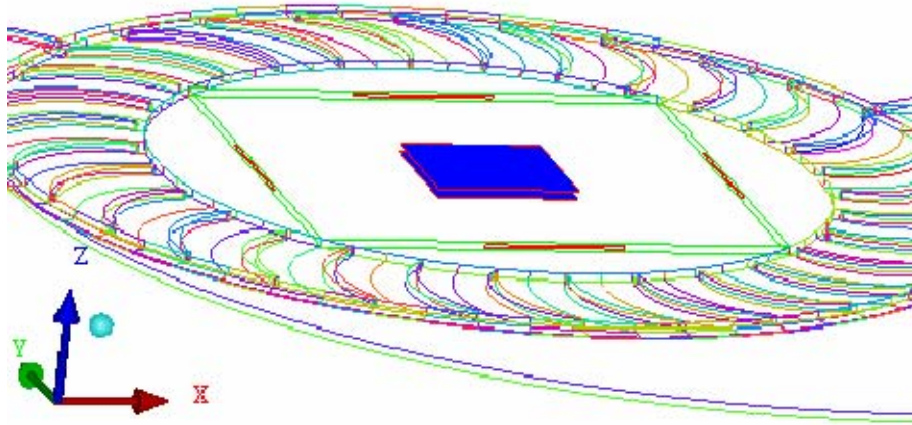




Figure D45:



**Blocking > Pre-Mesh Params**  **>Edge Param(s)** : It will open the *Meshing Parameters* window as shown in Figure D46. To create the mesh as shown in Figure D47, Select edges and enter their nodes as 36 or 18. Let **Mesh law** as BiGeometric and Check **Copy to parallel edges**.

After setting up the parameters on each edge, toggle **Pre-Mesh** from *tree widget*, then the mesh is created. Convert it to Unstruct Mesh before saving the file by following steps:

Use the **right mouse** and select **Pre-Mesh > Convert to Unstruct Mesh** in the model *Tree window*.

Figure D46: Meshing parameters window

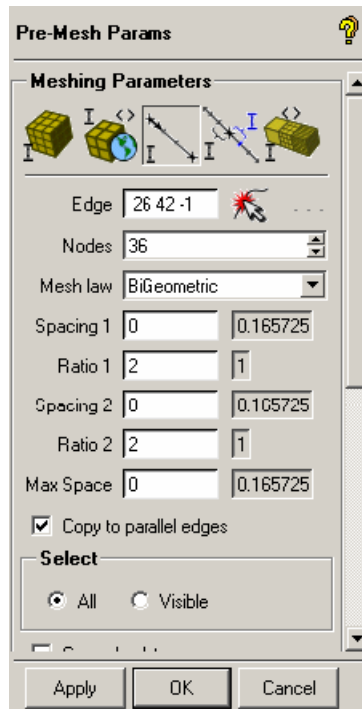
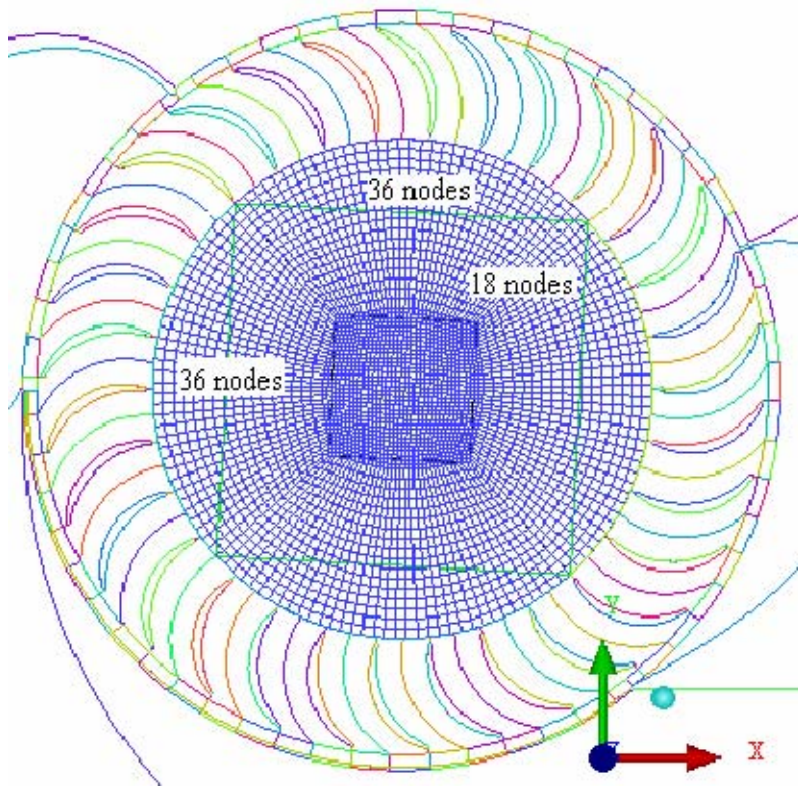


Figure D47: Completed center-circle mesh



#### 4.2. Blades Mesh

**Blocking > Create Block**  **> Initialize block** : Under **Initialize Block**, set **Type** to **2D Planer**. Name **Part** as **FLUID** and press **Apply** to create a single block.



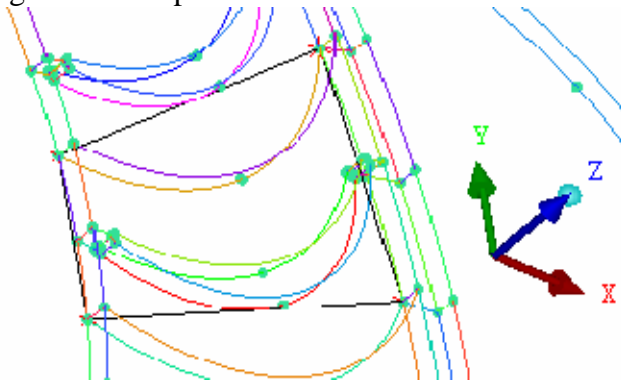
**Blocking > Associate**  **> Associate Vertex** : Toggle **ON Geometry > Points** and **Blocking > vertices > Numbers** from *tree widget*. Under **Entity**, select **Point**. And associate vertex to appropriate point as shown in Figure D48.

Figure D48: Moving vertices to points





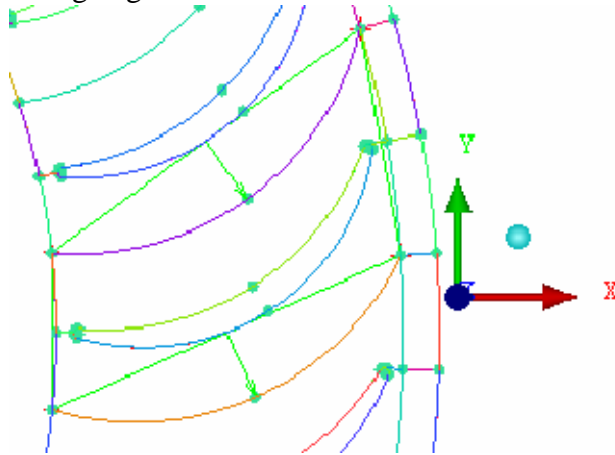
**Blocking > Associate**  **> Associate Edge to Curve**  : To associate block edges to appropriate curves as shown in Figure D49.

Figure D49: Associating edges to blade-curves





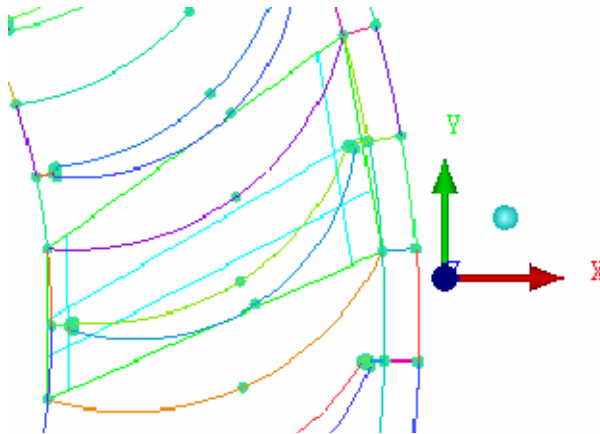
**Blocking > Split Block**  **> Split Block**  : Select the Top edge with the left mouse button, try to position the edge at the required position as shown in Figure D50, then press the middle mouse button to accept the selection. Create other split also.

Figure D50:



**Blocking > Associate**  **> Associate Edge to Curve**  : Associate inner block edges to appropriate blade-curves.

**Blocking > Associate**  **> Associate Vertex**  : Associate vertices at inner block to appropriate points at the blade.


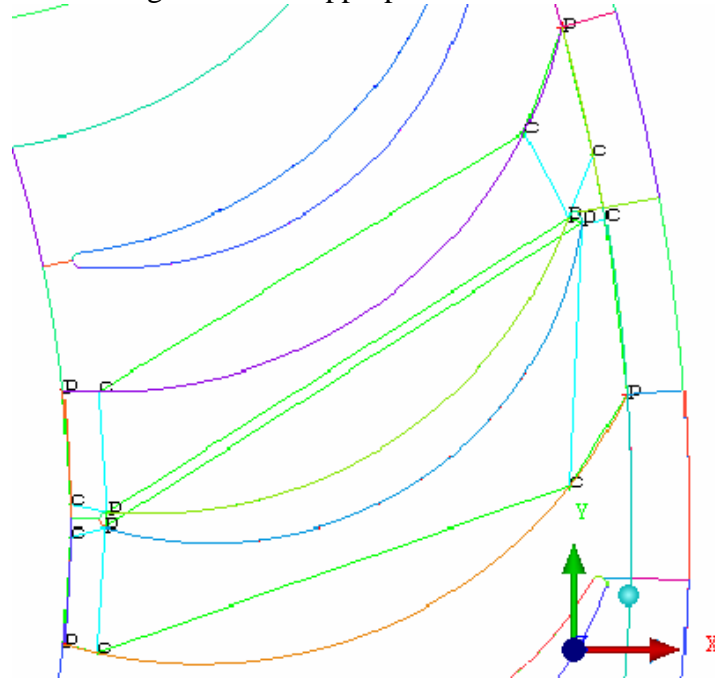
**Blocking > Move Vertex**  **> Move Vertex**  : Directly click on edges and move them to appropriate location as shown in Figure D51.

Figure D51: After moving vertices to appropriate location





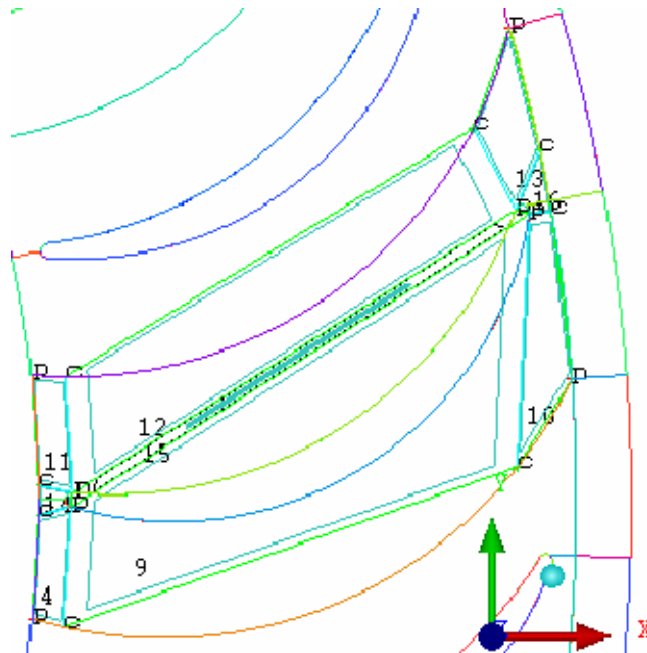
**Blocking > Split Block**  **> Ogrid Block**  : Set **Offset** to 0.3 and Check **Around Block(s)**. Select the block at center one as shown in Figure D52, and **Apply** to create O-grid block.

Figure D52: Select the block for the O-grid







**Blocking > Edit Edge**  **> Split Edge**  : To open *Edit Edge(s)* window as shown in Figure D53. Set **Split type** to Linear, then to split appropriate edges. Also use **Blocking > Move Vertex > Move Vertex** to move and arrange the edge as shown in Figure D54.

Figure D53: Edit edge window

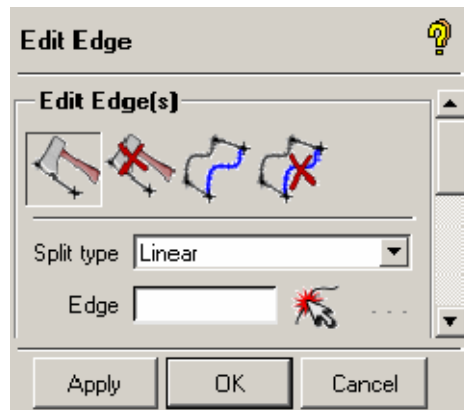
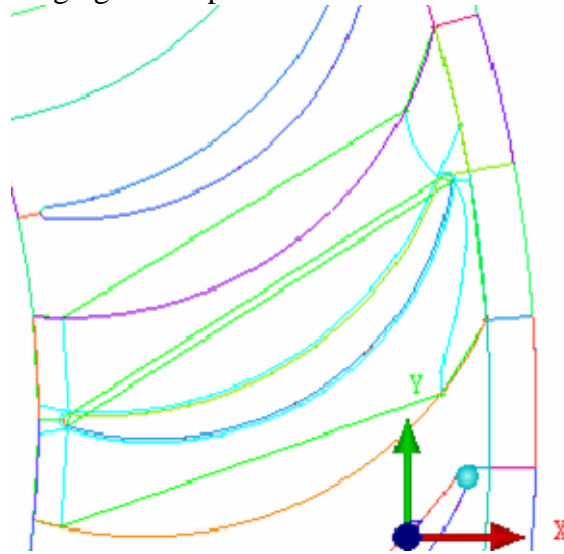




Figure D54: After arranging the shape of the block





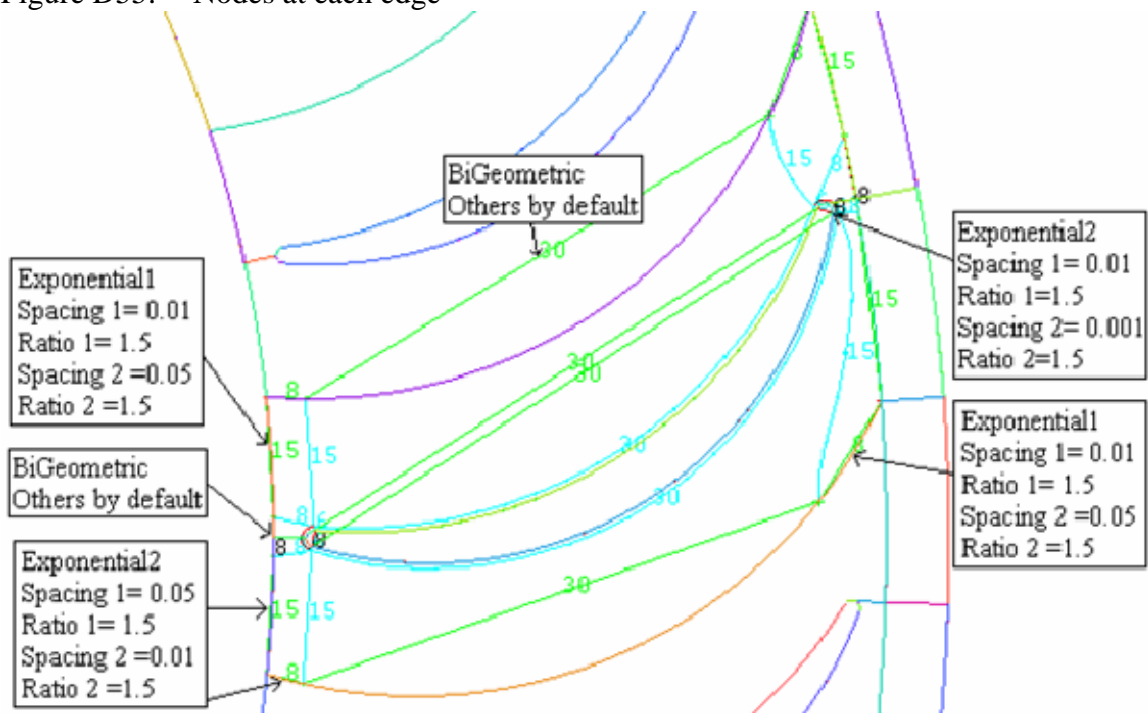
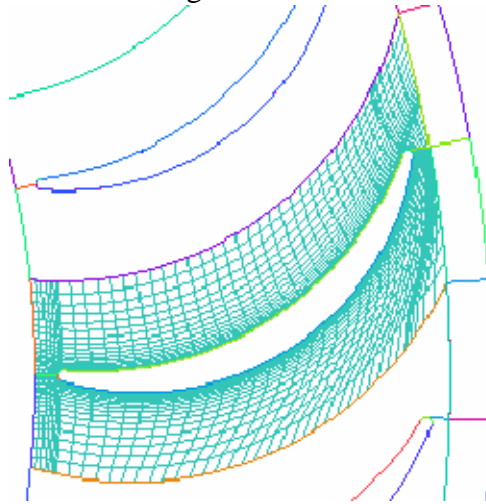
**Blocking > Pre-Mesh Params**  **>Edge Param(s)** : Set the number of nodes on each edge as shown in Figure D55. Check **Copy to parallel edges** and **Copy absolute**. Then press **Apply** to accept.

Figure D55: Nodes at each edge



After setting up the parameters on each edge, toggle **Pre-Mesh** from *tree widget*, then the mesh is created as shown in Figure D56.



Figure D56: Mesh created around single blade





Since this mesh is good enough, there are 29 blades remaining, it is suggested to rotate and copy this Block to other corresponding blades. Before the action, disassociate Edges and Vertices first.



Toggle **Pre-Mesh** from *tree widget* again to close the mesh

**Blocking > Associate**  **> Disassociate form Geometry** : To open *Disassociate* window as shown in Figure D57.

To select Edges, click on  and then select  from **Select Blocks** bar window. Press **Apply** to disassociate the edges.

**Blocking > Transform Blocks**  **> Rotate Blocks** : To open *selection* window.

To select **Rotating objects**, click on  and Click on  from **Select Blocks**. Check **Copy** and set **Number of copies** to **29**. Under **Rotation**, set on **Axis** to **Z**, and set **Angle** to **12**. Under **Center of Rotation**, set **Center Point** to **Origin**. Press **Apply** to rotate blocks.

**Blocking > Associate**  **> Associate Edge to Curve** : Re-associate edges to appropriate curves. And check if the nodes on edge are changed or not. If there are extra vertices, delete them by **Blocking > Edit Edge > Unsplit Edge**, under **Remove splits**, set **Single**.

After complete the previous step, to see how the meshes look like, toggle **Pre-Mesh** from *tree widget*, then the mesh is created as shown in Figure D59

Figure D57: Disassociate window

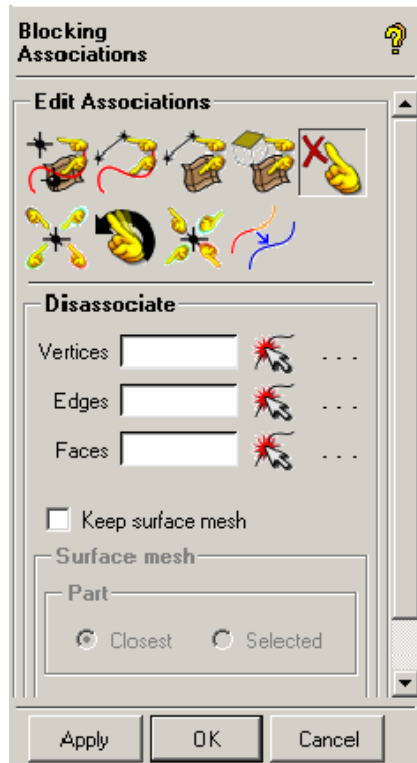


Figure D58: Rotation block selection window

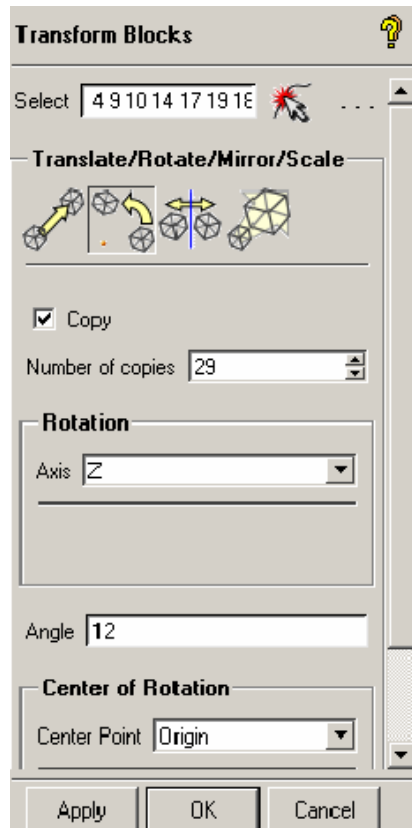
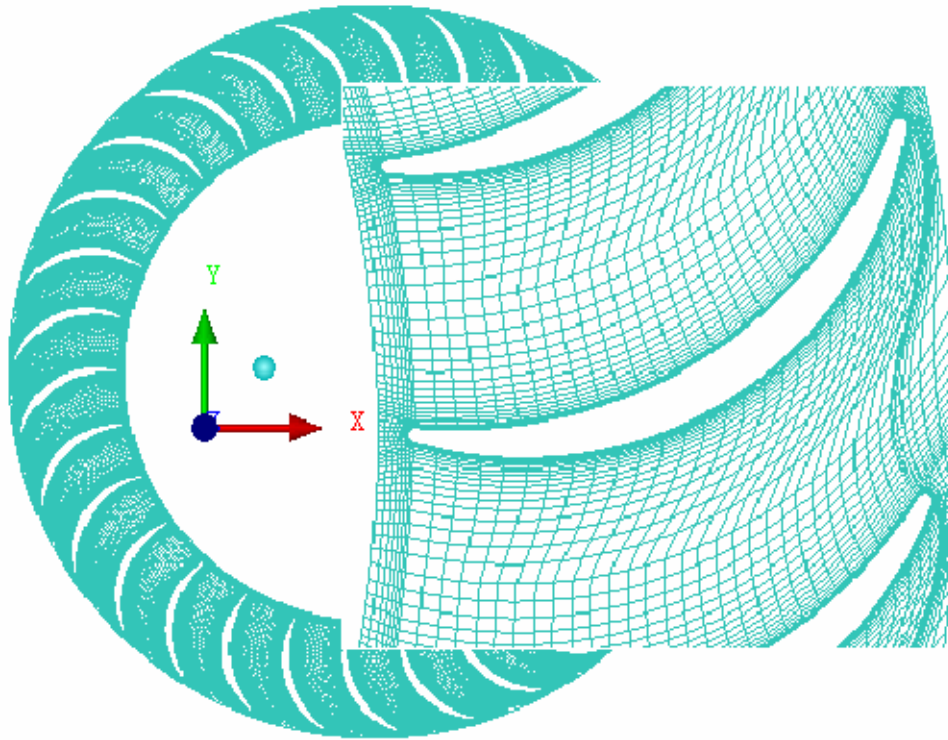


Figure D59: After 2D blade mesh creation





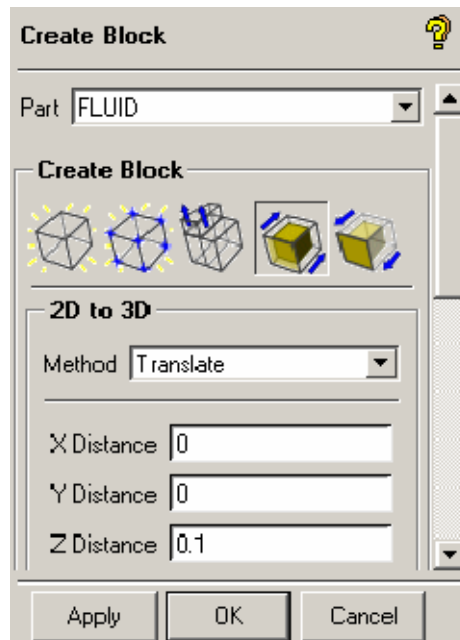
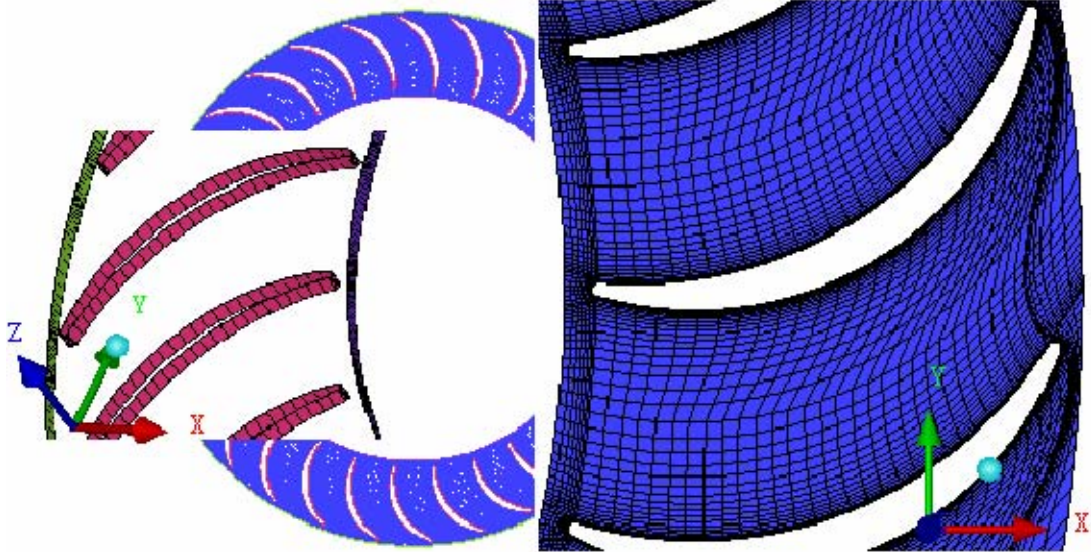
**Blocking > Create Block**  **> 2D to 3D** : To open *Method* window as shown in Figure D60. Set **Method** to **Translate** and **Z Distance** to **0.1**, others are **0**, press **Apply** to create 3D block.

Figure D60: Method window





Toggle **Pre-Mesh** from *tree widget* to see the quality of the meshes, disassociate edges and associate edges to curves if required. The final look of the meshes is shown in Figure 61.



Figure D61:



### 4.3 Inlet Section Mesh

**Blocking > Create Block**  **> Initialize block** : Under **Initialize Block**, set **Type** to **3D**. And select two opposite surfaces at inlet section. Name **Part** as **FLUID** and press **Apply** to create block.

**Blocking > Associate**  **> Associate Vertex** : Toggle **ON** Geometry > Points and Blocking > vertices > Numbers from *tree widget*. Under **Entity**, select **Point** and associate vertex to point.

**Blocking > Associate**  **> Associate Edge to Curve** : Associate edges to curves as shown in Figure D62.



**Blocking > Pre-Mesh Params**  **> Edge Param(s)** : Set the number of nodes on each edge as shown in Figure D63. Check **Copy to parallel edges** and **Copy absolute**. Then press **Apply** to accept.

Figure D62:

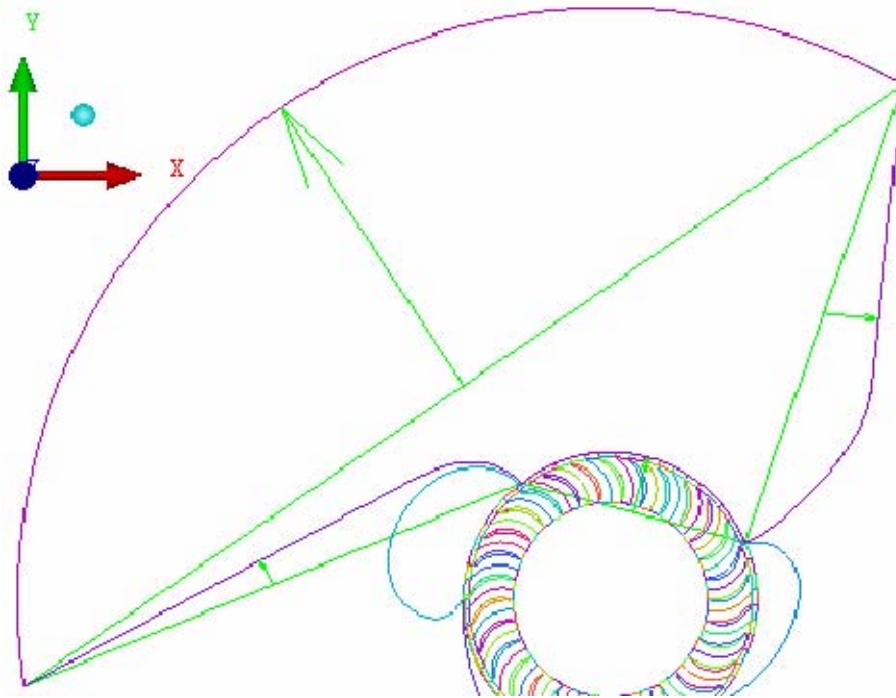
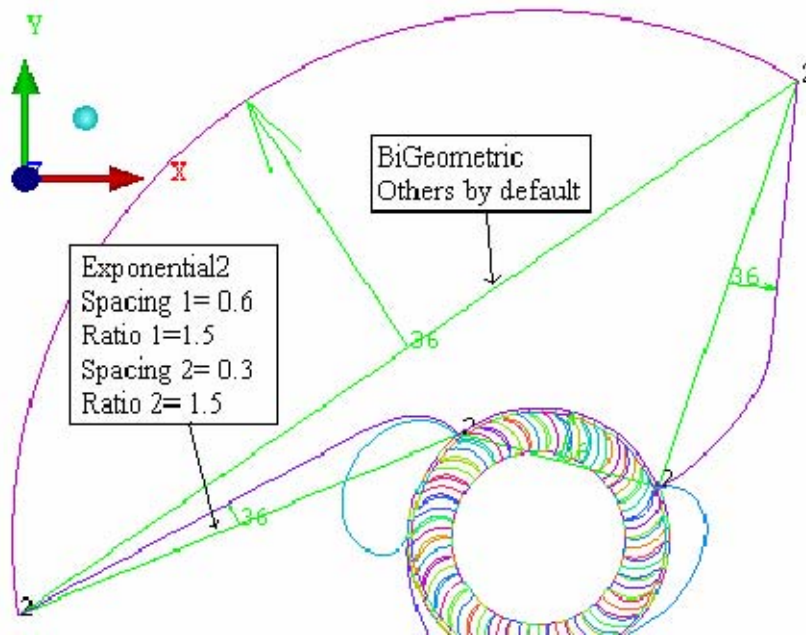


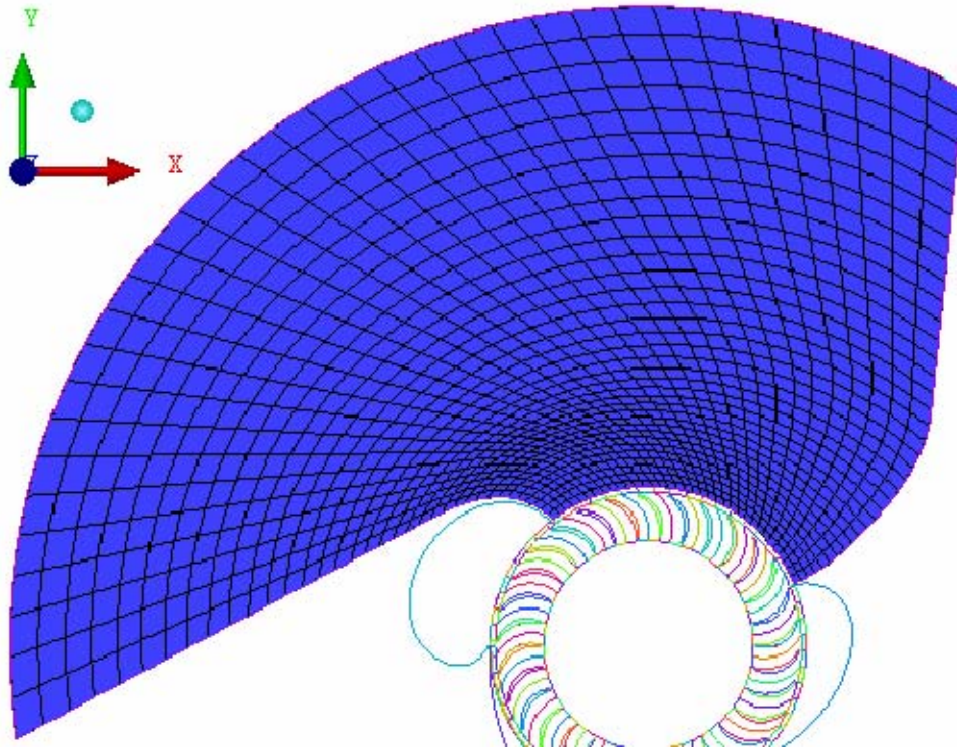
Figure D63:



After setting up the parameters on each edge, toggle **Pre-Mesh** from *tree widget*, then the mesh is created. **Convert it to Unstruct Mesh** before saving the file: Use the **right mouse** and select **Pre-Mesh > Convert to Unstruct Mesh** in the *model Tree window*. The final mesh at inlet section is shown in Figure D64.





Figure D64:



#### 4.4 Mesh between Inlet Section and Blades

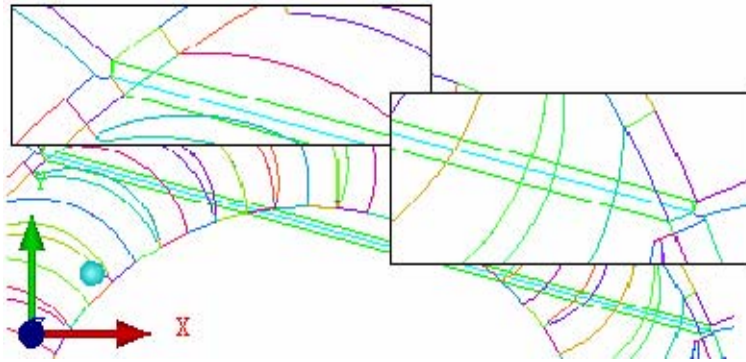
**Blocking > Create Block**  **> Initialize block** : Under **Initialize Block**, set **Type** to **2D Planer**. Name **Part** as **FLUID** and press **Apply** to create a single block.

**Blocking > Associate**  **> Associate Vertex** : Toggle **ON Geometry > Points** and **Blocking > vertices > Numbers** from *tree widget*. Under **Entity**, select **Point**. And associate vertices to appropriate points.

**Blocking > Split Block**  **> Split Block** : Split the left and right edge.

**Blocking > Associate**  **> Associate Vertex** : Associate vertices to appropriate points as shown in Figure D65.

Figure D65:





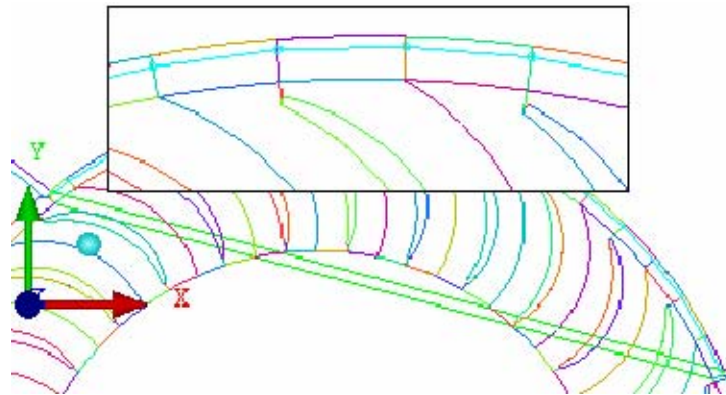
**Blocking > Edit Edge**  **> Split Edge**  : To open *Edit Edge(s)* window. Set **Split type** to **Linear**, then to split appropriate edges. Also use **Blocking > Move Vertex > Move Vertex** to move and arrange the edge as shown in Figure D66.

Figure D66:





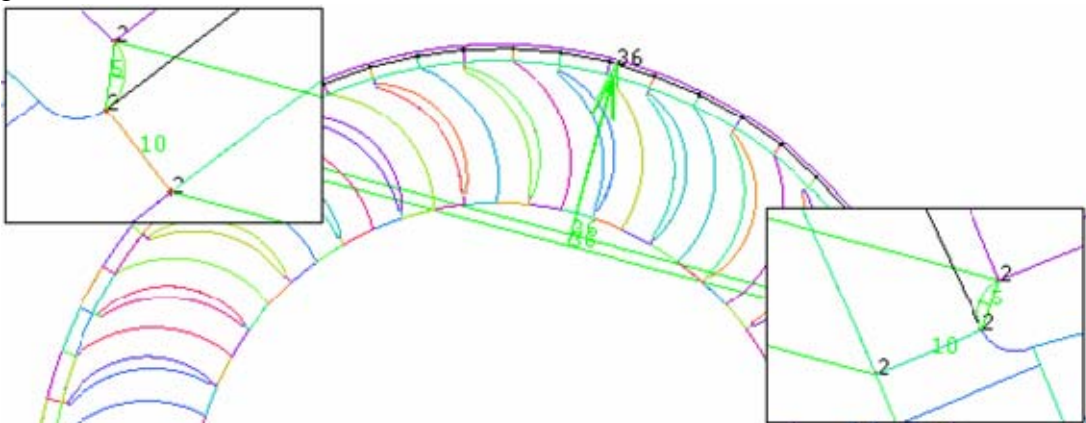


**Blocking > Pre-Mesh Params**  **>Edge Param(s)**  : Check **Copy to parallel edges** and **Copy absolute**. Set mesh parameter at each edge as shown in Figure D67.

Figure D67:





Toggle **Pre-Mesh** from *tree widget* and see mesh generated is proper or not. If it is acceptable, then follow the steps mentioned to generate meshes from 2D-3D.

**Blocking > Create Block**  **> 2D to 3D** : Set **Method** to **Translate** and **Z Distance** to **0.1**, others are **0**, press **Apply** to create 3D block.

Toggle **Pre-Mesh** from *tree widget* again to see the quality of the meshes. If edges are not associated properly, disassociate edges first, and then associate edges to curves.

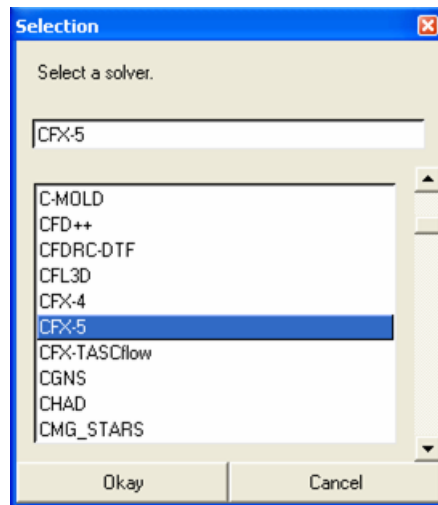
#### 4.4 Final Meshes

The meshes of other sections have to be generated by the previous procedure, either by 2D block to 3D block or 3D block only. It depends on the geometry. The meshes for each section are saved in different name. The final mesh was created by opening each mesh files and merging them into one mesh.

#### 4.5 Writing Output

**Output > Select solver:** Select the  (Select solver) to open the *Select* window as shown in Figure D68. Roll down to select CFX-5 solver.

Figure D68 Solver Selection Window




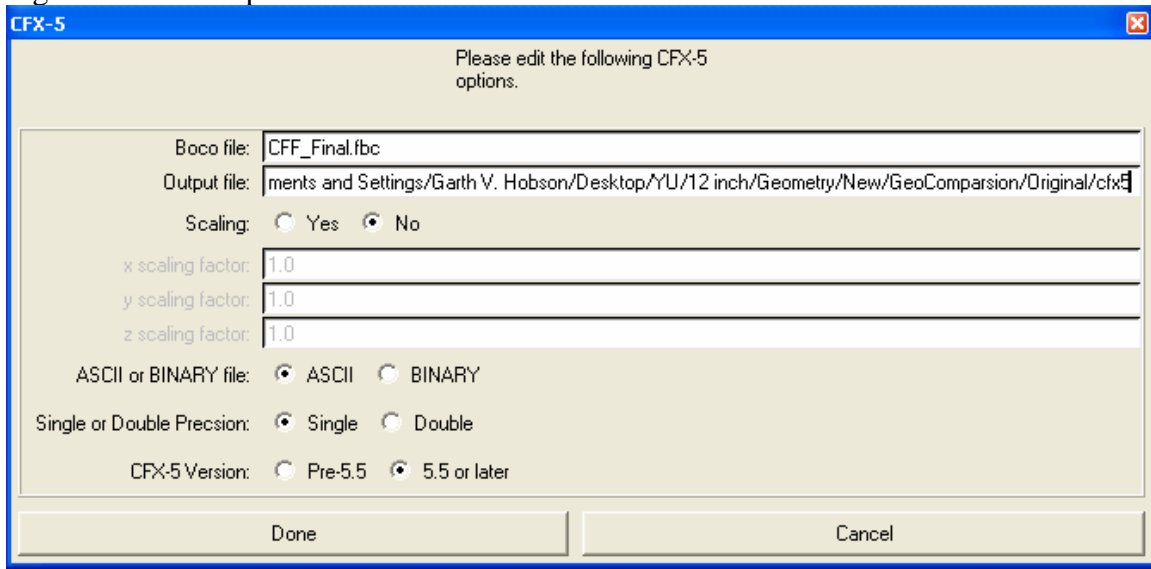
**Output > Write input:** Select the  (View input) to open the output file window as shown in Figure D69. In the directory, type the expected name such as \*.cfx5, then press Done.

Figure D69 Output File Window



The image shows a software window titled "CFX-5" with a standard Windows-style title bar (blue with a close button). The window's main area has a light beige background and contains the text "Please edit the following CFX-5 options." at the top. Below this, there are several configuration options, each with a label and a corresponding input field or radio button group. The options are: "Boco file:" with a text box containing "CFF\_Final.fbc"; "Output file:" with a text box containing a long file path; "Scaling:" with two radio buttons, "Yes" and "No", where "No" is selected; "x scaling factor:" with a text box containing "1.0"; "y scaling factor:" with a text box containing "1.0"; "z scaling factor:" with a text box containing "1.0"; "ASCII or BINARY file:" with two radio buttons, "ASCII" and "BINARY", where "ASCII" is selected; "Single or Double Precision:" with two radio buttons, "Single" and "Double", where "Single" is selected; and "CFX-5 Version:" with two radio buttons, "Pre-5.5" and "5.5 or later", where "5.5 or later" is selected. At the bottom of the window, there are two buttons: "Done" on the left and "Cancel" on the right.

CFX-5

Please edit the following CFX-5 options.

Boco file: CFF\_Final.fbc

Output file: ments and Settings/Garth V. Hobson/Desktop/YU/12 inch/Geometry/New/GeoComparsion/Original/cfx5

Scaling: ☐ Yes ☒ No

x scaling factor: 1.0

y scaling factor: 1.0

z scaling factor: 1.0

ASCII or BINARY file: ☒ ASCII ☐ BINARY

Single or Double Precision: ☒ Single ☐ Double


CFX-5 Version: ☐ Pre-5.5 ☒ 5.5 or later

Done Cancel



## APPENDIX E. PROCEDURES OF SETUP IN CFX-PRE

### I. Defining the Transient Simulation

#### A Creating a New Simulation


1. If required, launch CFX-Pre.
2. Select **File > New Simulation**.
3. Click **General**  in the New Simulation File Window.
4. If required, set the path location to a different folder.
5. Under **File name** type *CrossFlowFan*.
6. Click **Save**.
7. The **Mesh** tab is displayed.

#### B Importing a Mesh

1. Select **File > Import Mesh** or click **Import Mesh** .
2. Leave **Mesh Format** set to **ICEM CFD**.
3. In the bottom left of the screen, under **File** click **Browse** .
4. From the working directory, select *CrossFlowFan.cfx5*.
5. Click **Open**.
6. Set **Mesh Units** to **in**.
7. Click **Ok**.
8. The mesh is opened.

#### C Define the Simulation Type

In order to check if the solver will give convergent result or not, the first revolution is suggested and is set up as follows.


1. Select **Create > Flow Objects > Simulation Type**  from the main menu.  
**Simulation Type** appears
2. Set **Option** to **Transient**.
3. Under **Time Duration**, Set **Option** to **Total Time**, and **Total time** to **0.02 [s]**.
4. Under **Time Steps**, Set **Option** to **Timestep**, and **Timesteps** to **5.55555e-5 [s]**
5. Under **Initial Time**, Set **Option** to **Automatic with Value**, and **Time**

to **0** [s]


6. Click **Ok** to save the information.

## D Creating the Domain

### 1. To Create **Rotation Domain**

- a) Click **Create a Domain** .
- b) In **Create Domain**, set **Name** to *Rotation* and click **OK**.  
**Edit Domain: Rotation** is displayed
- c) On the **General Options** panel:
  - (1) Set **Location** to **Assembly, Assembly 3**.
  - (2) Leave **Domain Type** set to **Fluid Domain**.
  - (3) Set **Fluid List** to **Air Ideal Gas**.
  - (4) Leave **Coord Frame** set to **Coord 0**.
  - (5) Set **Reference Pressure** to **1** [atm].
  - (6) Under **Buoyancy**, leave **Option** set to **Non Buoyant**.
  - (7) Under **Domain Motion**, leave **Option** set to **Rotating**.
  - (8) Set **Angular Velocity** to **3000** [rev min<sup>-1</sup>].
  - (9) Turn on **Alternate Rotation Model**.
  - (10) Under **Axis Definition**, leave **Option** set to **Coordinate Axis**.
  - (11) Set **Rotation Axis** to **Global Z**.
- d) Click the **Fluid Models** tab and leave all options set to their respective defaults except:
  - (1) Under **Heat Transfer Model**, leave **Option** set to **Total Energy**.
  - (2) Turn on **Include Viscous Work Term**.
  - (3) Under **Turbulence Model**, leave **Option** set to **k-Epsilon**.
  - (4) Click **Ok** to create the domain.


### 2. To Create **Stationary Domain**



- a) Click **Create a Domain** .
- b) In **Create Domain**, set **Name** to *Stationary* and click **OK**.  
**Edit Domain: Stationary** is displayed
- c) On the **General Options** panel:

- (1) Set **Location** to **Assembly 2, Assembly 4, Assembly 5, Assembly 6, Assembly 7, Assembly 8, and Assembly 9, Assembly 10**.
  - (2) Leave **Domain Type** set to **Fluid Domain**.
  - (3) Set **Fluid List** to **Air Ideal Gas**.
  - (4) Leave **Coord Frame** set to **Coord 0**.
  - (5) Set **Reference Pressure** to 1 [atm].
  - (6) Under **Buoyancy**, leave **Option** set to **Non Buoyant**.
  - (7) Under **Domain Motion**, leave **Option** set to **Stationary**.
- d) Click the **Fluid Models** tab and leave all options set to their respective defaults except:
- (1) Under **Heat Transfer Model**, leave **Option** set to **Total Energy**.
  - (2) Turn on **Include Viscous Work Term**.
  - (3) Under **Turbulence Model**, leave option set to **k-Epsilon**.
- e) Click **Ok** to create the domain.


## E Creating the Boundary Conditions

### 1. To Create the Inlet Boundary Condition


- a) Click **Create a Boundary Condition**  on the main toolbar.
- b) Set **Name** to *Inlet* and **Domain** to **Stationary**, then **Ok**.  
**Edit Boundary** is displayed.
- c) On the **Basic Settings** panel, set:
  - (1) **Boundary Type** to **Opening**
  - (2) **Location** to **INLET**
- d) Click the **Boundary Details** tab, then:
  - (1) Under **Flow Regime**, leave **Option** set to **Subsonic**.
  - (2) Under **Mass and Momentum**, set **Option** to **Pressure and Direction (Stable)**.
  - (3) Set **Relative Pressure** to 0 [Pa]
  - (4) Under **Flow Direction**, set **Option** to **Normal to Boundary Condition**.
  - (5) Under **Turbulence**, set **Option** to **Medium (Intensity = 5%)**.

- (6) Under **Heat Transfer**, set **Option** to **Static Temperature**.
  - (7) Set **Static Temperature** to 300 [K].
- e) Click **Ok** to create the boundary.
2. To Create the **Outlet Boundary Condition**
  - a) Click **Create a Boundary Condition** .
  - b) Set **Name** to *Outlet* and **Domain** to **Stationary**, then **Ok**.  
**Edit Boundary** is displayed.
  - c) On the **Basic Settings** panel, set:
    - (1) **Boundary Type** to **Opening**
    - (2) **Location** to **OUTLET**
  - d) Click the **Boundary Details** tab, then:
    - (1) Under **Flow Regime**, leave **Option** set to **Subsonic**.
    - (2) Under **Mass and Momentum**, set **Option** to **Static Pressure**.
    - (3) Set **Relative Pressure** to **0** [Pa]
    - (4) Under **Flow Direction**, set **Option** to **Normal to Boundary Condition**.
    - (5) Under **Turbulence**, set **Option** to **Medium (Intensity = 5%)**.
    - (6) Under **Heat Transfer**, set **Option** to **Static Temperature**.
    - (7) Set **Static Temperature** to 300 [K].
  - e) Click **Ok** to create the boundary condition.
3. To Create the **Symmetry Boundary Condition**  
First create the symmetric boundary condition in **Stationary Domain**.
  - a) Click **Create a Boundary Condition** .
  - b) Set **Name** to *SYM1* and **Domain** to **Stationary**, then **Ok**.  
**Edit Boundary** is displayed.
  - c) On the **Basic Settings** panel, set:
    - (1) **Boundary Type** to **Symmetry**
    - (2) **Location** to **SYM1 C, SYM1 D, SYM1 E, SYM1 F, SYM1 G, SYM1 H, SYM1 I, SYM1 J**
  - d) Click **Ok** to create the boundary condition.


Similarly create the opposite symmetric boundary condition.

- a) Click **Create a Boundary Condition** .
- b) Set **Name** to *SYM2* and **Domain** to **Stationary**, then **Ok**.  
**Edit Boundary** is displayed.
- c) On the **Basic Settings** panel, set:
  - (1) **Boundary Type** to **Symmetry**
  - (2) **Location** to **SYM2 C, SYM2 D, SYM2 E, SYM2 F, SYM2 G, SYM2 H, SYM2 I, SYM2 J**
- d) Click **Ok** to create the boundary condition.


Then create the symmetric boundary condition in **Rotation Domain**.


- a) Click **Create a Boundary Condition** .
- b) Set **Name** to *SYM3* and **Domain** to **Rotation**, then **Ok**.  
**Edit Boundary** is displayed.
- c) On the **Basic Settings** panel, set:
  - (1) **Boundary Type** to **Symmetry**
  - (2) **Location** to **SYM1 A, SYM1 B**
- d) Click **Ok** to create the boundary condition.

Similarly create the opposite symmetric boundary condition


- a) Click **Create a Boundary Condition** .
- b) Set **Name** to *SYM4* and **Domain** to **Rotation**, then **Ok**.  
**Edit Boundary** is displayed.
- c) On the **Basic Settings** panel, set:
  - (1) **Boundary Type** to **Rotation**
  - (2) **Location** to **SYM2 A, SYM2 B**
- d) Click **Ok** to create the boundary condition.

## 2. To Create the Wall Boundary Condition




- a) Click **Create a Boundary Condition** .
- b) Set **Name** to *Wall* and **Domain** to **Stationary**, then **Ok**.  
**Edit Boundary** is displayed.
- c) On the **Basic Settings** panel, set:
  - (3) **Boundary Type** to **Wall**
  - (4) **Location** to **Wall**

- d) Click the **Boundary Details** tab, then:
    - (1) Under **Wall Influence On Flow**, leave **Option** set to **No Slip**.
    - (2) Under **Wall Roughness**, set **Option** to **Smooth Wall**.
    - (3) Under **Heat Transfer**, set **Option** to **Adiabatic**.
  - e) Click **Ok** to create the boundary.
3. To Create the Blades Boundary Condition
- a) Click **Create a Boundary Condition** .
  - b) Set **Name** to *Blades* and **Domain** to **Rotation**, then **Ok**.  
**Edit Boundary** is displayed.
  - c) On the **Basic Settings** panel, set:
    - (1) **Boundary Type** to **Wall**
    - (2) **Location** to **Blades**
  - d) Click the **Boundary Details** tab, then:
    - (1) Under **Wall Influence On Flow**, leave **Option** set to **No Slip**.
    - (2) Under **Wall Roughness**, set **Option** to **Smooth Wall**.
    - (3) Under **Heat Transfer**, set **Option** to **Adiabatic**.
  - e) Click **Ok** to create the boundary.

## D Creating Domain Interface


1. To create the interface between **Rotation Domain** and **Stationary Domain**
  - a) Click **Create a Domain Interface**  on the main toolbar.
  - b) Leave **Name** by default (Domain Interface 1), then **Ok**.  
**Edit Domain Interface** is displayed.
  - c) On **Basic Setting**, set **Interface Type** to **Fluid Fluid**.
  - d) Under **Side 1**, set **Domain** to **Rotation**.
  - e) Set **Region List 1** to **OUTERCIRCLE A**.
  - f) Under **Side 2**, set **Domain** to **Stationary**.
  - g) Set **Region List 2** to **OUTERCIRCLE B, OUTERCIRCLE C, OUTERCIRCLE D, OUTERCIRCLE E**.
  - h) On **Interface Models**, under **Frame Change**, set **Option** to **Transient Rotor Stator**.
  - i) Under **Pitch Change**, set **Option** to **Automatic**.




- j) Click **Ok** to create the interface.
2. To create **the interface** in **Stationary Domain**
- a) Domain Interface 2
- (1) Click **Create a Domain Interface**  on the main toolbar.
  - (2) Leave **Name** by default, then **Ok**.
  - (3) On **Basic Setting**, set Interface Type to Fluid Fluid.
  - (4) Under **Side 1**, set Domain to **All Domain**.
  - (5) Set **Region List 1** to **INTERCIRCLE A**.
  - (6) Under **Side 2**, set Domain to **Stationary**.
  - (7) Set **Region List 2** to **INTERCIRCLE B**.
  - (8) On **Interface Models**, under **Frame Change**, set **Option** to **None**.
  - (9) Under **Pitch Change**, set **Option** to **Automatic**.
  - (10) Click **Ok** to create the interface.
- b) Domain Interface 3
- (1) Click **Create a Domain Interface**  on the main toolbar.
  - (2) Leave **Name** by default, then **Ok**.
  - (3) On **Basic Setting**, set Interface Type to Fluid Fluid.
  - (4) Under **Side 1**, set Domain to **Stationary**.
  - (5) Set **Region List 1** to **INT 1 A**.
  - (6) Under **Side 2**, set Domain to **Stationary**.
  - (7) Set **Region List 2** to **INT 1 B**.
  - (8) On **Interface Models**, under **Frame Change**, set **Option** to **None**.
  - (9) Under **Pitch Change**, set **Option** to **Automatic**.
  - (10) Click **Ok** to create the interface.
- c) Domain Interface 4
- (1) Click **Create a Domain Interface**  on the main toolbar.
  - (2) Leave **Name** by default, then **Ok**.
  - (3) On **Basic Setting**, set **Interface Type** to Fluid Fluid.

- (4) Under **Side 1**, set **Domain** to **Stationary**.
- (5) Set **Region List 1** to **INT 2 A**.
- (6) Under **Side 2**, set **Domain** to **Stationary**.
- (7) Set **Region List 2** to **INT 2 B**.
- (8) On **Interface Models**, under **Frame Change**, set **Option** to **None**.
- (9) Under **Pitch Change**, set **Option** to **Automatic**.
- (10) Click **Ok** to create the interface.


d) Domain Interface 5

- (1) Click **Create a Domain Interface**  on the main toolbar.
- (2) Leave **Name** by default, then **Ok**.
- (3) On **Basic Setting**, set **Interface Type** to **Fluid Fluid**.
- (4) Under **Side 1**, set **Domain** to **Stationary**.
- (5) Set **Region List 1** to **INT 3 A**.
- (6) Under **Side 2**, set **Domain** to **Stationary**.
- (7) Set **Region List 2** to **INT 3 B**.
- (8) On **Interface Models**, under **Frame Change**, set **Option** to **None**.
- (9) Under **Pitch Change**, set **Option** to **Automatic**.
- (10) Click **Ok** to create the interface.


e) Domain Interface 6

- (1) Click **Create a Domain Interface**  on the main toolbar.
- (2) Leave **Name** by default, then **Ok**.
- (3) On **Basic Setting**, set **Interface Type** to **Fluid Fluid**.
- (4) Under **Side 1**, set **Domain** to **Stationary**.
- (5) Set **Region List 1** to **INT 4 A**.
- (6) Under **Side 2**, set **Domain** to **Stationary**.
- (7) Set **Region List 2** to **INT 4 B**.
- (8) On **Interface Models**, under **Frame Change**, set **Option** to **None**.
- (9) Under **Pitch Change**, set **Option** to **Automatic**.
- (10) Click **Ok** to create the interface.


f) Domain Interface 7


- (1) Click **Create a Domain Interface**  on the main toolbar.
- (2) Leave **Name** by default, then **Ok**.
- (3) On **Basic Setting**, set **Interface Type** to **Fluid Fluid**.
- (4) Under **Side 1**, set **Domain** to **Stationary**.
- (5) Set **Region List 1** to **INT 5 A**.
- (6) Under **Side 2**, set **Domain** to **Stationary**.
- (7) Set **Region List 2** to **INT 5 B**.
- (8) On **Interface Models**, under **Frame Change**, set **Option** to **None**.
- (9) Under **Pitch Change**, set **Option** to **Automatic**.
- (10) Click **Ok** to create the interface.

g) Domain Interface 8


- (1) Click **Create a Domain Interface**  on the main toolbar.
- (2) Leave **Name** by default, then **Ok**.
- (3) On **Basic Setting**, set **Interface Type** to **Fluid Fluid**.
- (4) Under **Side 1**, set **Domain** to **Stationary**.
- (5) Set **Region List 1** to **INT 6 A**.
- (6) Under **Side 2**, set **Domain** to **Stationary**.
- (7) Set **Region List 2** to **INT 6 B**.
- (8) On **Interface Models**, under **Frame Change**, set **Option** to **None**.
- (9) Under **Pitch Change**, set **Option** to **Automatic**.
- (10) Click **Ok** to create the interface.

h) Domain Interface 9


- (1) Click **Create a Domain Interface**  on the main toolbar.
- (2) Leave **Name** by default, then **Ok**.
- (3) On **Basic Setting**, set **Interface Type** to **Fluid Fluid**.
- (4) Under **Side 1**, set **Domain** to **Stationary**.
- (5) Set **Region List 1** to **INT 7 A**.

- (6) Under **Side 2**, set Domain to Stationary.
  - (7) Set **Region List 2** to **INT 7 B**.
  - (8) On **Interface Models**, under **Frame Change**, set **Option** to **None**.
  - (9) Under Pitch Change, set Option to Automatic.
  - (10) Click **Ok** to create the interface.
- i) Domain Interface 10
- (1) Click **Create a Domain Interface**  on the main toolbar.
  - (2) Leave **Name** by default, then **Ok**.
  - (3) On **Basic Setting**, set **Interface Type** to **Fluid Fluid**.
  - (4) Under **Side 1**, set **Domain** to **Stationary**.
  - (5) Set **Region List 1** to **INT 8 A**.
  - (6) Under **Side 2**, set **Domain** to **Stationary**.
  - (7) Set **Region List 2** to **INT 8 B**.
  - (8) On **Interface Models**, under **Frame Change**, set **Option** to **None**.
  - (9) Under **Pitch Change**, set **Option** to Automatic.
  - (10) Click **Ok** to create the interface.

## E Setting Initial Values


1. Select **Create > File Objects > Global Initialisation** from the main menu bar, or click **Global Initialisation** .
2. Under **Initial Conditions**, set **Velocity Type** to **Cartesian**. Other **Options** are set to **Automatic**.
3. Turn on **Turbulence Eddy Dissipation** and leave **Option** set to **Automatic**.
4. Click **Ok** to set the initialisation details.

## F Setting Solver Control

1. Select **Create > Flow Objects > Solver Control**, or click **Solver Control**  to open **Solver Control**.
2. Under **Advection Scheme**, leave **Option** set to **Upwind**.
3. Under **Transient Scheme**, set **Option** to **Second Order Backward Euler**.
4. Under **Convergence Control**, Set **Max. Iter. Per Timestep** to **3**.

5. Under **Convergence Criteria**, set **Residual Type** to **RMS**.
6. Set **Residual Target** to **1e-8**.
7. Click on **Conservation Target**, set **Conservation Target** to **1e-8**.
8. Click **Ok** to set the solver parameter.


#### G Setting Output Control

1. Select **Create > Flow Objects > Output Control**  from the main menu bar.
2. Click the **Transient Results** tab.
3. Click **Add New Item** and then click **Ok** to accept the default name for the object.
4. Under **Transient Results 1**:
  - a) Set **Option** to **Full**.
  - b) Leave **File Compression Level** to **Default**
  - c) Turn on **Time Interval**, then set **Time Interval** to **0.001 [s]**
5. Click **Ok**.

#### H To start without initial value for all domain

1. **Create > Expert Objects > Expert Parameters** from the main menu bar.
2. Click **I/O Control** tab.
3. Under **Transient Model and I/O**, Turn on **transient initialization override**, and then set **transient initialization** to **t**.
4. Click **Ok** to accept.

#### I Writing the Solver (.def) File

1. Click **Write Solver (.def) File** . **Definition File** is created.
2. **File name** *CrossFlowFan.def* is created automatically.
3. Leave **Operation** set to **Start Solver Manger**.
4. Leave **Report Summary of Interface Connections** and Quit **CFX-Pre** turned on.
5. Click **Ok**.
6. Click **Yes** when asked if you want to save the CFX file.

#### J Obtaining a Solution to this Transient Problem

1. When **CFX-Pre** closes and then **CFX-Solver Manager** will start, click on **Start Run** to start the run.


The residual plots for seven variables will appear: P-Mass, U-Mom, V-Mom, W-Mom, H-Energy, E-Diss.K, and K-TurbKE.

2. When **CFX-Solver** finishes, the completion message will appear. Click **Ok** to acknowledge it.
3. Click **Post-Process Results** to see the result.

## II. Continuing the Transient Run

Based on the result obtained previously, more revolutions are required and are set as follows.


### A Redefine the Simulation Type

1. Select **Create > Flow Objects > Simulation Type**  from the main menu.
2. Set **Option** to **Transient**.
3. Under **Time Duration**, Set **Option** to **Total Time**, and **Total time** to **0.1 [s]**.

This will provide 5 revolutions totally.

4. Under **Time Steps**, Set **Option** to **Timestep**, and **Timesteps** to **3.33333e-5 [s]**
5. Under **Initial Time**, Set **Option** to **Automatic**.  
The initial time will be set automatically from the previous run.
6. Click **Ok** to save the information.

### B Writing the Solver (.def) File

1. Click **Write Solver (.def) File** .  
**Definition File** is created.
2. Change **File name** to *CrossFlowFan1.def*  
It is happened that solver cannot run due to definition file existed already, so changing the file name somewhat is suggested.
3. Leave **Operation** set to **Start Solver Manger**.
4. Leave **Report Summary of Interface Connections** and Quit **CFX-Pre** turned on.
5. Click **Ok**.
6. Click **Yes** when asked if you want to save the CFX file.  
**CFX-Solver Manager** window appears after clicking **Yes**.

### C Obtaining the Solution to this Transient Problem

1. On **Run Definition tab**, leave **Definition File** to *CrossFlowFan1.def*.

2. Set **Initial Values File** to **CrossFlowFan\_001.res.** which is obtained from the previous run.
3. Click on **Start Run** to continue the run.
4. When **CFX-Solver** finishes, the completion message will appear. Click **Ok** to complete it.
5. Click **Post-Process Results** to see the result.

Note: Sometimes there is a message popping up, it says, “the memory could not be read,” even there is enough memory in the computer. The procedure to ignore this problem is as follows:

1. Click on **Solver** tab
2. Under **Solver Memory**, Set **Memory Allocation Factor** to 2 or more. Hence the memory shared for CFX-Solver will be increased.

THIS PAGE INTENTIONALLY LEFT BLANK



## LIST OF REFERENCES

1. Naval Air System Command contract N00019-74-C-0434, *Multi-Bypass Ratio Propulsion System Technology Development*, Vol. I-III, Vought Systems Division, LTV Aerospace Corporation, 24 July 1975.
2. Gossett D. H., " Investigation of Cross Flow Fan Propulsion for a lightweight VTOL Aircraft", Master's Thesis, Department of Aeronautics and Astronautics, Naval Postgraduate School, Monterey, California, March 2000.
3. Seaton, M. S., " Performance Measurements, Flow visualization, and Numerical Simulation of a CrossFlow Fan", Master's Thesis, Department of Aeronautics and Astronautics, Naval Postgraduate School, Monterey, California, March 2003.
4. Cheng, W. T., " Experimental and Numerical Analysis of a CrossFlow Fan", Master's Thesis, Department of Aeronautics and Astronautics, Naval Postgraduate School, Monterey, California, December 2003.
5. Kummer, J. D., Dang. T.Q., "High-Lift Propulsive Airfoil with Integrated Crossflow Fan", *Journal of Aircraft* Vol. 43, No.4, July-August 2006.
6. Schreiber, C. W., " Effect of Span Variation on the Performance of a Cross Flow Fan", Master's Thesis, Department of Mechanical and Astronautical Engineering, Naval Postgraduate School, Monterey, California, June 2006.
7. Shreeve, R. P., "Report of the Testing of a Hybrid (Radial to Axial) Compressor", Naval Postgraduate School, Department of Aeronautics, NPS-57Sf73112A, November 1973.
8. White, F.M., *Fluid Mechanics*, fifth edition, McGraw-Hill, San Francisco, 2003.

THIS PAGE INTENTIONALLY LEFT BLANK

## INITIAL DISTRIBUTION LIST

1. Defense Technical Information Center  
Ft. Belvoir, Virginia
2. Dudley Knox Library  
Naval Postgraduate School  
Monterey, California
3. Prof. Max F. Platzer  
Department of Mechanical and Aeronautical Engineering  
Naval Postgraduate School  
Monterey, California
4. Prof. Garth V. Hobson  
Department of Mechanical and Aeronautical Engineering  
Naval Postgraduate School  
Monterey, California
5. Prof. Raymond P. Shreeve  
Department of Mechanical and Aeronautical Engineering  
Naval Postgraduate School  
Monterey, California
6. Dr. Anthony Gannon  
Department of Mechanical and Aeronautical Engineering  
Naval Postgraduate School  
Monterey, California
7. Prof. Jr-Ming Miao  
Department of Mechanical Engineering  
National Defense University  
Taiwan
8. Captain Huai-Te Yu  
Naval Postgraduate School  
Monterey, California